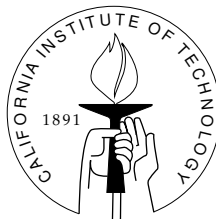


EERL 2003-03  
Earthquake Engineering  
Research Laboratory

# **FRAME3D - A Program for Three-Dimensional Nonlinear Time-History Analysis of Steel Buildings: User Guide**

by  
Swaminathan Krishnan



California Institute of Technology  
Pasadena, California

2003

© 2003

Swaminathan Krishnan

All Rights Reserved

## Summary

FRAME3D is a program for the three-dimensional nonlinear analysis of steel buildings. It aims to overcome the computational challenges posed by full 3D analysis of buildings subject to earthquake ground motion. The element library consists of a plastic hinge beam element, an elastofiber beam element, a panel zone element, a 4-noded diaphragm element to model floor slabs, and an elastic translational/rotational spring element to model foundations and supports. The program utilizes a Newton-Raphson iteration strategy applied to an implicit Newmark time-integration scheme to solve the nonlinear equations of motion at each time-step. Geometric nonlinearity and shear deformation are included in the formulation. This document serves as a User Guide to the program. All the input and output variables encountered by the user are described here along with brief descriptions of the various types of elements. In addition, 2 examples illustrating the capabilities and usage of the program are presented. Finally a glossary of all the variables is alphabetically listed at the end of the document for the user's convenience.

# Contents

<b>Summary</b>	<b>iii</b>
<b>1 Introduction</b>	<b>1</b>
<b>2 Program Outline</b>	<b>2</b>
2.1 Description of Model . . . . .	3
2.1.1 Panel Zone Element . . . . .	3
2.1.2 Beam Element . . . . .	3
2.1.3 Diaphragm Element . . . . .	3
2.1.4 Coordinate Systems . . . . .	5
2.1.5 Structure Degrees of Freedom . . . . .	6
2.2 Equations of Motion . . . . .	7
<b>3 Element Library</b>	<b>11</b>
3.1 Panel Zone Element . . . . .	11
3.1.1 Material Models for Panel Shear . . . . .	13
3.2 Plastic Hinge Beam Element . . . . .	16
3.2.1 Degrees of Freedom, and Nodal Forces and Moments . . . . .	19
3.2.2 Internal Forces and Moments . . . . .	19
3.2.3 Material Nonlinearity . . . . .	20
3.3 Elastofiber Beam Element . . . . .	21
3.3.1 Degrees of Freedom of a Segment, and Nodal Forces and Moments .	24
3.3.2 Internal Forces and Moments in a Segment . . . . .	26
3.4 Material Model for Fiber Axial Stress-Strain . . . . .	26
3.5 Elastic 4-Noded Diaphragm Element . . . . .	27
3.5.1 Degrees of Freedom, and Nodal Forces . . . . .	28
3.5.2 Internal Stresses in a Diaphragm Element . . . . .	30
3.6 Elastic Translational or Rotational Spring Element . . . . .	30

<b>4</b>	<b>Input Data</b>	<b>31</b>
4.1	Input File for005 . . . . .	31
4.1.1	Heading . . . . .	34
4.1.2	Main Control Data . . . . .	34
4.1.3	Integration and Iteration Data . . . . .	38
4.1.4	Nodal Data . . . . .	44
4.1.5	Spring Element Data . . . . .	47
4.1.6	Beam Element Data . . . . .	48
4.1.7	Categories for Elastofiber Element Fiber Area Reduction . . . . .	52
4.1.8	Seed for Random Number Generation for Elastofiber Element Fiber Fracture Stress Computation . . . . .	54
4.1.9	Categories for Elastofiber Element Fiber Fracture Stress Prob. Dis- tribution . . . . .	55
4.1.10	Panel Zone Element Data . . . . .	57
4.1.11	Diaphragm Element Control Data . . . . .	59
4.1.12	Diaphragm Element Material Data . . . . .	60
4.1.13	Diaphragm Element Connectivity Data . . . . .	61
4.1.14	Diaphragm Element Output Control Data . . . . .	63
4.1.15	Time History Output Control Data . . . . .	64
4.1.16	Ground Acceleration Scaling Factors . . . . .	67
4.1.17	Deformed Shape Output Control Data . . . . .	68
4.2	Input File for009 . . . . .	69
4.2.1	Labeling . . . . .	69
4.2.2	Beam Section Data . . . . .	69
4.3	Input Files for090, for091 and for092 . . . . .	70
4.3.1	Heading . . . . .	70
4.3.2	Acceleration Data Points . . . . .	71
<b>5</b>	<b>Program Output</b>	<b>72</b>
5.1	File for006 . . . . .	72
5.2	File RTH . . . . .	78
5.3	File COORD . . . . .	78

5.4	File DEFCOORD . . . . .	78
5.5	File BEAM . . . . .	79
5.6	File PLSTR . . . . .	79
5.7	File BMPLROT . . . . .	80
5.8	File PZPLROT . . . . .	80
5.9	File PLSTRSS . . . . .	80
5.10	File PLSTRSMAX . . . . .	81
<b>6</b>	<b>Examples</b>	<b>82</b>
6.1	Large Deflection of an Elastic Cantilever Beam . . . . .	82
6.2	2 Story Building with 2 Bays in Each Principal Direction . . . . .	89
<b>A</b>	<b>Glossary of Input &amp; Output Variables</b>	<b>106</b>
	<b>Bibliography</b>	<b>114</b>

## List of Figures

2.1	Element Arrangement in Frame Model, Showing Nodes, Attachment Points, and Coordinate Systems . . . . .	4
2.2	Panel zone shear caused by beam and column moments at the joint . . . . .	5
2.3	Degrees of Freedom of Panel Zone Element . . . . .	8
3.1	Panel Zone Element . . . . .	12
3.2	Shear Stress-Strain Model for a Bilinear Panel Zone Element . . . . .	14
3.3	Shear Stress-Strain Model for a Linear-Quadratic Panel Zone Element . . . . .	16
3.4	DOF of the Plastic Hinge Beam Element Showing Nodal Translations and Rotations and Nodal Forces and Moments . . . . .	18
3.5	Sign Convention for Internal Forces and Couples in a Plastic Hinge Beam Element . . . . .	20
3.6	$P - M_{pY'} - M_{pZ'}$ Interaction Diagrams used for the Plastic Hinge Element . . . . .	21
3.7	Layout of the 3-Segment Elastofiber Element . . . . .	23
3.8	DOF of a Segment of the Elastofiber Beam Element Showing Nodal Translations and Rotations and Nodal Forces and Moments . . . . .	25
3.9	Stress-Strain Model for a Typical Fiber in a Fiber Segment of an Elastofiber Element . . . . .	27
3.10	DOF of the Diaphragm Element Showing Nodal Translations and Rotations and Nodal Forces and the Isoparametric Mapping from a Square Parent Element . . . . .	29
4.1	Beam/Column Element Orientation Angle . . . . .	50
6.1	Large Deflection of a Cantilever Beam . . . . .	82
6.2	Large Deflection of a Cantilever Beam: Input file, <i>for005</i> . . . . .	83
6.3	Large Deflection of a Cantilever Beam: Input file, <i>for009</i> . . . . .	84
6.4	Large Deflection of a Cantilever Beam: Output file, <i>for006</i> . . . . .	85
6.5	Large Deflection of a Cantilever Beam: Output file, <i>for006</i> (Contd.) . . . . .	86

6.6	Large Deflection of a Cantilever Beam: Output file, <i>for006</i> (Contd.) . . . .	87
6.7	Large Deflection of a Cantilever Beam: Output file, <i>for006</i> (Contd.) . . . .	88
6.8	Isometric View of 2 Story Building . . . . .	90
6.9	Plan View of 2 Story Building . . . . .	91
6.10	2-Story Building: Input file, <i>for005</i> . . . . .	92
6.11	2-Story Building: Input file, <i>for005</i> (Contd.) . . . . .	93
6.12	2-Story Building: Input file, <i>for009</i> . . . . .	94
6.13	2-Story Building: Output file, <i>for006</i> . . . . .	95
6.14	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	96
6.15	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	97
6.16	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	98
6.17	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	99
6.18	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	100
6.19	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	101
6.20	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	102
6.21	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	103
6.22	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	104
6.23	2-Story Building: Output file, <i>for006</i> (Contd.) . . . . .	105



## List of Tables

3.1	FRAME3D Element Library . . . . .	11
4.1	Input Files for FRAME3D Analysis . . . . .	31
4.2	File for005: List of Data Blocks . . . . .	33
5.1	Output Files from FRAME3D Analysis . . . . .	72
5.2	File for006: List of Output Sections . . . . .	73

# Chapter 1 Introduction

FRAME3D is a comprehensive 3D building analysis program ideally suited for analyzing steel-framed buildings subject to earthquake motions. A variety of elements suitable for nonlinear modeling of structural elements in a building are provided. A brief outline of the program with its capabilities is given in Chapter 2. The element library with a brief description of each element is given in Chapter 3. The format for the user input files is provided in Chapter 4 along with the description of all input variables. Program output is described in Chapter 5. Two examples illustrating the usage and capabilities of the program are provided in Chapter 6. Finally, an alphabetically ordered glossary of input and output variables is provided for the user's convenience in Appendix A.

For detailed information on the implementation of the program, the user is referred to [10, 11, 12].

This program has been successfully used, with reasonable computing resources, to analyze 20-story buildings subject to strong ground motion from earthquakes causing significant nonlinearity in structural behavior [10]. It is believed that it can be used for the analysis of 40-60 story buildings with reasonable computing resources.

## Chapter 2 Program Outline

The implementation of the program is described in detail in [10] and [12]. The following are the key features and capabilities of the program:

1. Stepped nonlinear static 3D analysis of buildings.
2. Nonlinear dynamic 3D time-history analysis of buildings.
3. Element library includes inelastic beam elements, inelastic panel zone elements, elastic diaphragm elements to model floors and elastic spring elements to recover reactions.
4. P-Delta effects are automatically included (geometric updating). This also enables large deformation problems to be solved.
5. Shear deformation is included in beam element formulation.
6. Effect of axial force on moment capacity of beam elements is included.
7. Stiffness matrix is stored in banded form thus minimizing storage requirements.
8. Damping matrix is of the proportional Rayleigh damping type. It could be mass proportional or stiffness proportional or a combination of the two.
9. Option to iterate with the initial elastic stiffness matrix or with the tangent stiffness matrix or a combination of both.
10. Time-histories of nodal displacements, element forces and inelastic deformation can be written out and can be plotted using standard plotting programs.
11. Element peak forces and inelastic deformations can be written out and maps of plastic deformation of various frames can be created using a program such as matlab to gain an understanding of the damage observed in the structure during severe earthquakes.
12. The deformed shape of the structure in the form of updated nodal coordinates can be written out at any desired time interval. Using a program such as matlab, an animation of the structure's shaking during an earthquake can then be created.

## 2.1 Description of Model

This work deals with the modeling of 3D frame structures consisting of grids of beams and columns. The structural model is comprised of three element types: panel zone elements, beam elements for beams and columns, and diaphragm elements (See Figure 2.1).

### 2.1.1 Panel Zone Element

This element models nonlinear shear deformation in the region of the joint where the beams and columns intersect. The joint region consists of a length of column within the depth of the connecting beams. The shear deformation is due primarily to opposing moments from the beams and columns at the joint caused by the frame being subjected to lateral loads (Figure 2.2). Each panel zone element is associated with a global node  $J$ ,  $K$ , etc. at the center of the joint where the global degrees of freedom (DOF) are defined. Each panel zone element consists of two orthogonal panels ① and ② which always remain planar and orthogonal. Edges of these panels contain attachment points  $a$ ,  $b$ ,  $c$ , and  $d$ , where beams attach and  $e$  and  $f$  on the top and bottom where columns attach. Details of the panel zone element are presented in Chapter 3.

### 2.1.2 Beam Element

This element is used to model beams and columns. Two types of beam elements have been developed: plastic hinge type and elastofiber type. The plastic hinge element has 2 nodes with local node numbers 1 and 2, while the elastofiber element consists of 3 segments with 4 nodes. The exterior nodes at the ends of the element are numbered 1 and 2, while the interior nodes are numbered 3 and 4. Both types of beam elements consider nonlinear behavior for flexural and axial deformations. Details of the plastic hinge element and the elastofiber element are given in Chapter 3.

### 2.1.3 Diaphragm Element

This element is used to model the in-plane stiffness of floor slabs. It is essentially a 4-noded plane-stress element connecting to global nodes  $J$ ,  $K$ ,  $L$ , and  $M$ . Details of the diaphragm element are presented in Chapter 3.

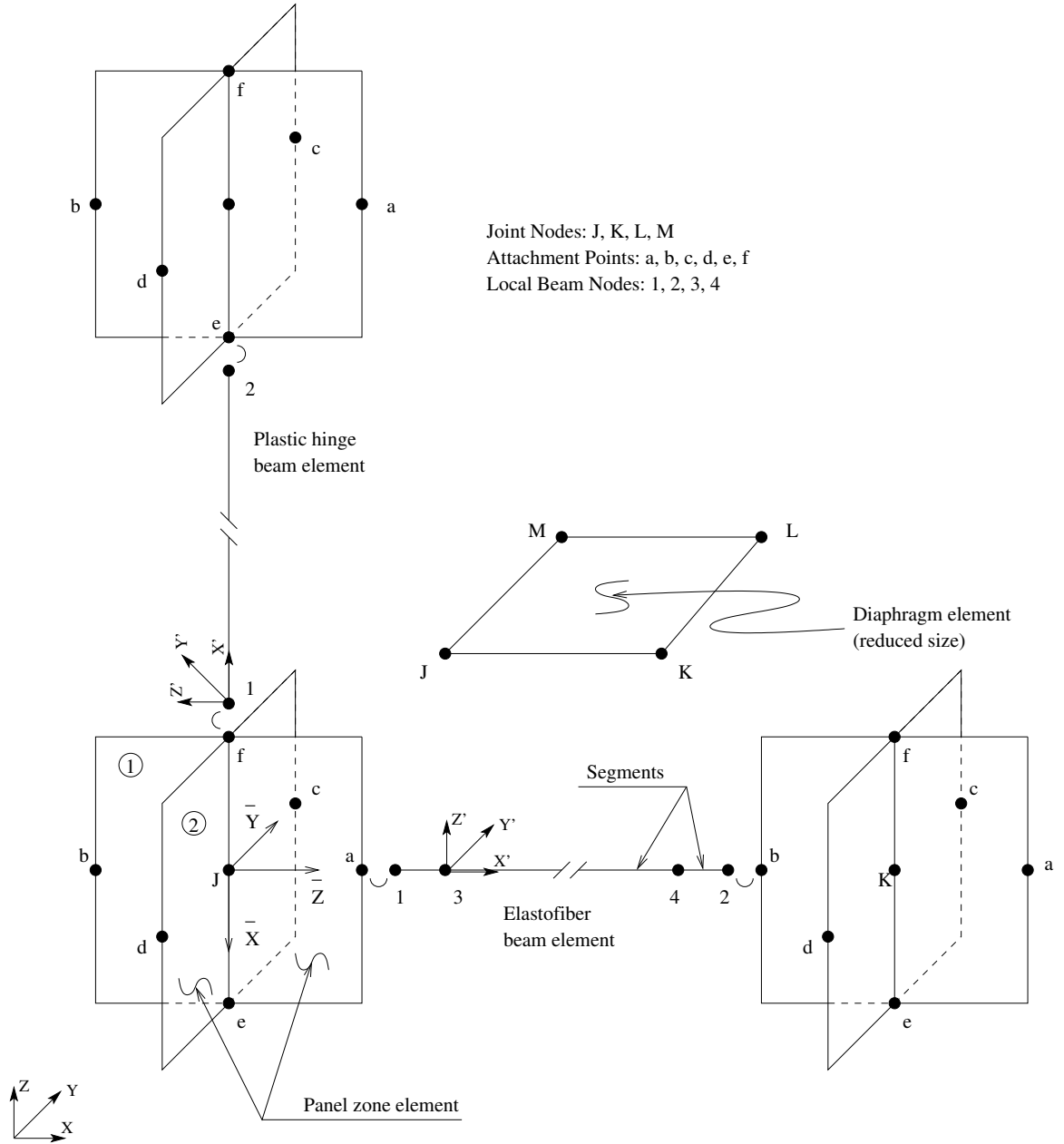


Figure 2.1: Element Arrangement in Frame Model, Showing Nodes, Attachment Points, and Coordinate Systems

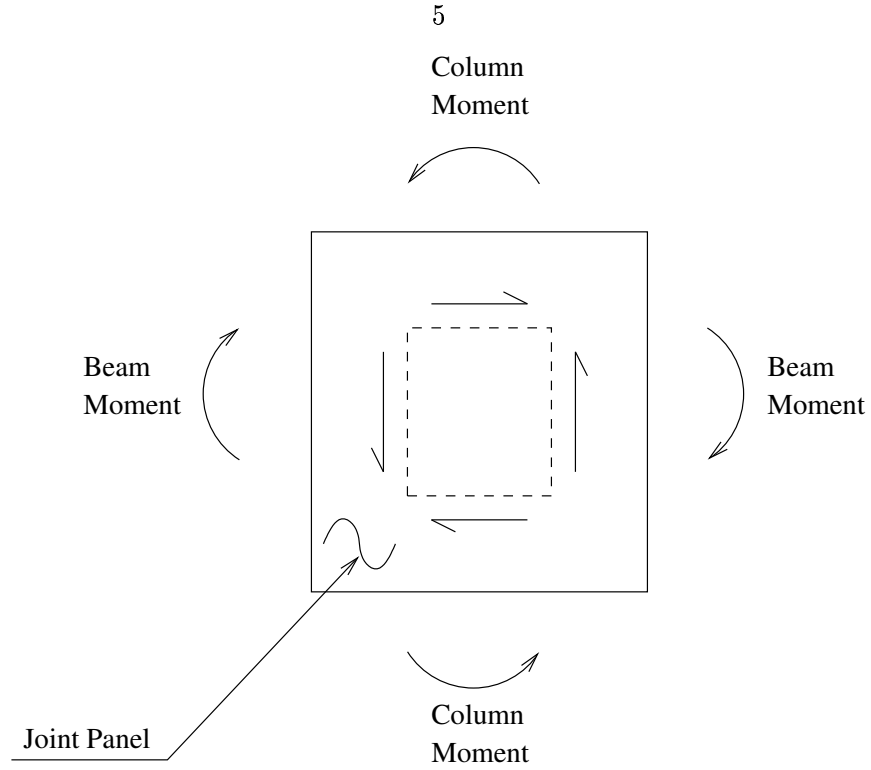


Figure 2.2: Panel zone shear caused by beam and column moments at the joint

#### 2.1.4 Coordinate Systems

The following are the various coordinate systems used in the structural model:

1.  $XYZ$ : The global coordinate system,  $XYZ$ , is the fixed coordinate system that is used to define the structure in space. The coordinates of the nodes defined by the user are in this coordinate system. In general it is recommended that  $Z$  axis be taken to be vertical (direction of gravity) and the  $X$  and  $Y$  axes be oriented in the two principal directions of the building.
2.  $\bar{X}\bar{Y}\bar{Z}$ : This coordinate system is local to the panel zone element.  $\bar{X}$  lies along the line connecting attachment points  $e$  and  $f$ , in the direction of a vector running from  $f$  to  $e$ .  $\bar{X} - \bar{Z}$  is the plane of panel ① and  $\bar{X} - \bar{Y}$  is the plane of panel ②. This coordinate system translates and rotates with the panel zone element.
3.  $X'Y'Z'$ : This coordinate system is local to the plastic hinge beam element.  $X'$  lies along the longitudinal axis of the beam running from node 1 to node 2.  $Y' - Z'$  is the cross-sectional plane of the beam.  $Y'$  is the major principal axis of the cross-section while  $Z'$  is the minor axis. This coordinate system translates and rotates with the

beam element.

4.  $X'Y'Z'$ : This coordinate system is local to each segment of the elastofiber beam element. Its orientation is similar to the plastic hinge beam element except that each segment has its own  $X'Y'Z'$  system. Each translates and rotates with the corresponding segment of the element.
5.  $X'Y'$ : This coordinate system is local to the plane of the diaphragm element with nodes  $J$ ,  $K$ ,  $L$ , and  $M$ . Its origin lies at the center of the element.  $X'$  axis is in the direction of the vector running from node  $K$  to  $L$  and  $Y'$  is perpendicular to  $X'$  and in the plane of the element.

### 2.1.5 Structure Degrees of Freedom

The global degrees of freedom (DOF) are associated only with the nodes  $J$ ,  $K$ , etc. at the centers of joints. The structural model consists of 8 global DOF at each node,  $J$ . These DOF are listed below:

1.  $U_{JX}$ : Translation in the  $X$  direction.
2.  $U_{JY}$ : Translation in the  $Y$  direction.
3.  $U_{JZ}$ : Translation in the  $Z$  direction.
4.  $\theta_{J\bar{X}}$ : Rotation of panel zone element as a rigid body about the  $\bar{X}$  axis.
5.  $\theta_{J\bar{Y}}^B$ : Rotation of the line  $e - f$  about the  $\bar{Y}$  axis. It corresponds to deformation of panel ① into the shape of a parallelogram (Figure 2.3a). The  $B$  indicates that the rotating panel edges are connected to beams.
6.  $\theta_{J\bar{Y}}^C$ : Rotation of the line  $a - b$  about the  $\bar{Y}$  axis. It corresponds to deformation of panel ① into the shape of a parallelogram (Figure 2.3b). The  $C$  indicates that the rotating panel edges are connected to columns. Note that  $\theta_{J\bar{Y}}^B$  and  $\theta_{J\bar{Y}}^C$  together accomodate a rigid rotation of panel ① about  $\bar{Y}$  plus its shear deformation.
7.  $\theta_{J\bar{Z}}^B$ : Rotation of the line  $e - f$  about the  $\bar{Z}$  axis. It corresponds to deformation of panel ② into the shape of a parallelogram (Figure 2.3c). The  $B$  indicates that the rotating panel edges are connected to beams.

8.  $\theta_{J\bar{Z}}^C$ : Rotation of the line  $c-d$  about the  $\bar{Z}$  axis. It corresponds to deformation of panel ② into the shape of a parallelogram (Figure 2.3d). The  $C$  indicates that the rotating panel edges are connected to columns. Note that  $\theta_{J\bar{Z}}^B$  and  $\theta_{J\bar{Z}}^C$  together accomodate a rigid rotation of panel ② about  $\bar{Z}$  plus its shear deformation.

Panel zone elements contribute stiffness directly to the four global DOF,  $\theta_{J\bar{Y}}^B$ ,  $\theta_{J\bar{Y}}^C$ ,  $\theta_{J\bar{Z}}^B$ , and  $\theta_{J\bar{Z}}^C$ . Beam elements are formulated in terms of 3 translational and 3 rotational DOF at each of the local nodes (1 and 2 for plastic hinge element and 1, 2, 3, and 4 for elastofiber element). However, none of these DOF appear in the global equations. Apprpriate stiffness terms are assembled into the global DOF through transformation matrices based on the geometry of the panel zone elements at nodes 1 and 2.

In addition to material nonlinearity, geometric stiffness (effect of axial force on flexural stiffness) is included with the beam elements. In addition, as the building deforms, locations of global nodes, local beam nodes and attachment points are updated, as are all the local coordinate systems. This ensures that  $P - \Delta$  effects are accounted for accurately.

Mass is lumped into the global translational DOF at nodes  $J$ ,  $K$ , etc. Damping is assumed to be of the Rayleigh type (stiffness and mass proportional). Building foundation is assumed to be rigid.

## 2.2 Equations of Motion

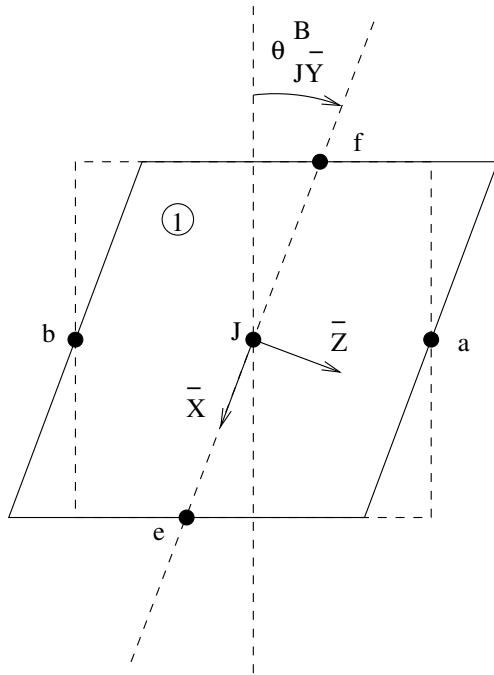
The matrix equation of motion of the building [3] as a function of time,  $t$ , is:

$$[M] \left\{ \ddot{U}(t) \right\} + [C] \left\{ \dot{U}(t) \right\} + \{R(t)\} = \{f_g\} - [M] [r] \left\{ \ddot{U}_g(t) \right\} \quad (2.1)$$

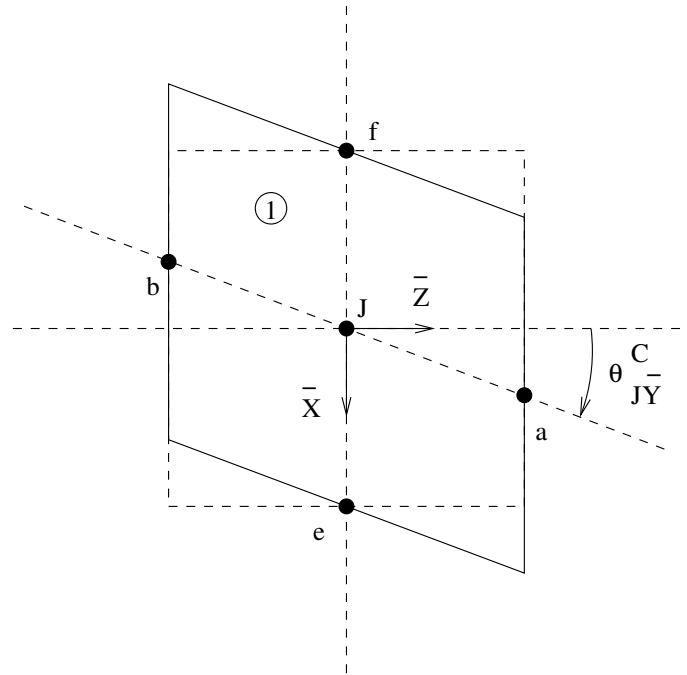
In the above,

1.  $\{U(t)\}$  = Vector of global displacements at time,  $t$ , comprising of the set of 8 global translations and rotations at each node,  $J$ ,  $K$ , etc.
2.  $\left\{ \dot{U}(t) \right\}$ ,  $\left\{ \ddot{U}(t) \right\}$  = Vectors of nodal velocities and accelerations respectively corresponding to the 8 global translations and rotations at each node.
3.  $[M]$  = Structure mass matrix. The mass in the structural system is lumped into the translational degrees of freedom at the joint nodes rendering the mass matrix

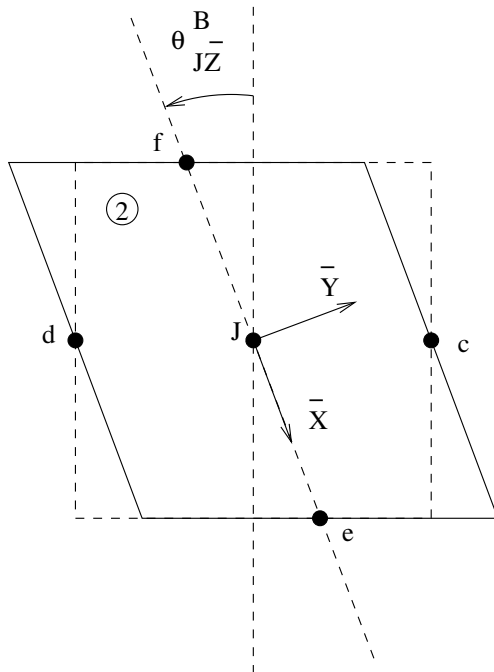




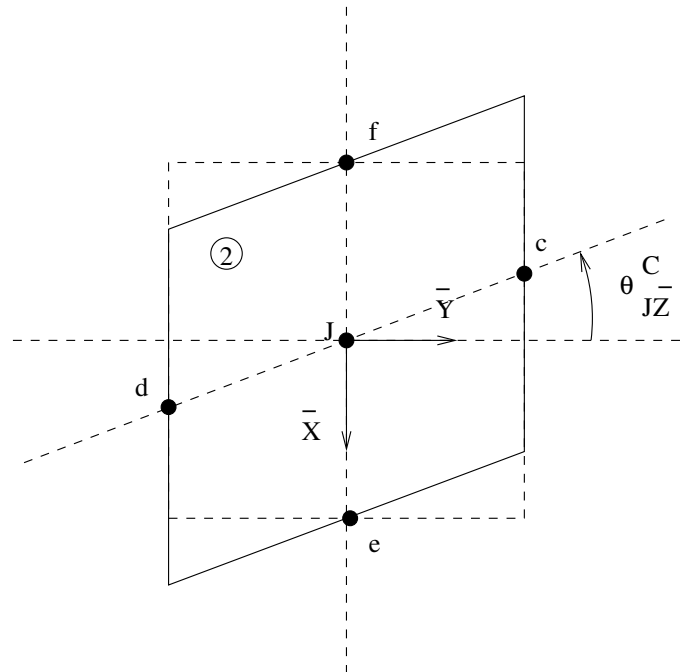
(a)  $\theta_{J\bar{Y}}^B$  DOF



(b)  $\theta_{J\bar{Y}}^C$  DOF



(c)  $\theta_{J\bar{Z}}^B$  DOF



(d)  $\theta_{J\bar{Z}}^C$  DOF

Figure 2.3: Degrees of Freedom of Panel Zone Element

diagonal. Further, the rotary inertia of the lumped nodal masses is neglected, and so the only non-zero terms of the mass matrix are the diagonal terms corresponding to the translational degrees of freedom.

4.  $[C]$  = Structure damping matrix.

Damping is assumed to be of the Rayleigh type (stiffness and mass proportional). Thus, the damping matrix is computed as

$$[C] = \alpha_0 [M] + \alpha_1 [K] \quad (2.2)$$

where  $a_0$  and  $a_1$  are user defined proportionality constants. The initial elastic stiffness matrix is used in the above computation.

In the linear elastic case, where modal superposition is valid, if Rayleigh damping is assumed, the modal damping in the  $i^{th}$  mode can be written in terms of the corresponding modal frequency as

$$\zeta_i = \frac{1}{2} \left( \alpha_0 \omega_i + \frac{\alpha_1}{\omega_i} \right) \quad (2.3)$$

If we plot  $\omega_i$  vs  $\zeta_i$ , the curve is found to be reasonably flat in the intermediate frequency range. Assuming that the structure exhibits approximately equal damping values in all the modes, two frequencies,  $\omega$  and  $R\omega$ , are picked covering the frequency range of interest and a desired amount of damping. Then  $a_0$  and  $a_1$  can be computed as

$$\alpha_0 = \frac{2\zeta\omega R}{1 + R} \quad (2.4a)$$

$$\alpha_1 = \frac{2\zeta}{\omega(1 + R)} \quad (2.4b)$$

5.  $\{R(t)\}$  = Vector of stiffness forces corresponding to the configuration,  $\{U(t)\}$ . It is computed considering all material and geometric nonlinear effects.
6.  $\{f_g\}$  = Vector of static gravity loads for which a static analysis is performed first.
7.  $\{\ddot{U}_g(t)\}$  = Vector consisting of two horizontal components ( $X$  and  $Y$ ) and one vertical component ( $Z$ ) of the free-field ground acceleration at time  $t$ . The ground motion is assumed uniform.

8.  $[r] = A$  3-column matrix of zeroes except for ones in the first, second, and third columns in the positions corresponding to the  $X$ ,  $Y$ , and  $Z$  DOF, respectively.
9. The base DOF are fixed and are not included.

## Chapter 3 Element Library

The element library of program, FRAME3D, is listed in Table 3.1.

Table 3.1: FRAME3D Element Library

S.No.	Element
1	Panel zone element.
2	Plastic-hinge beam element.
3	Elastofiber beam element.
4	Elastic diaphragm element.
5	Elastic translational/rotational spring element.

### 3.1 Panel Zone Element

Each joint is modeled by a panel zone element consisting of a length of column within the depth of the connecting beams. This implies that one column, the associated column of the joint (ACJ) runs continuously through the height of the joint. The panel zone element is an idealization of the joint region of this column. It consists of two rectangular panels which are perpendicular to each other, panel ① in the  $\bar{X} - \bar{Z}$  plane and panel ② in the  $\bar{X} - \bar{Y}$  plane, forming a cruciform section (Figure 3.1). The thicknesses of all web plates and web doubler plates of the ACJ are combined to form the thickness of panel ①,  $t_p^{①}$ , while the thicknesses of all flange plates of the ACJ are combined to form the thickness of panel ②,  $t_p^{②}$ . The depth,  $D$ , of the joint is taken to be equal to the depth (dimension along the minor axis,  $Z'$ ) of the ACJ. The width,  $W$ , of the joint is taken to be equal to the width (dimension along the major axis,  $Y'$ ) of the ACJ. The height,  $H$ , of the joint is taken to be equal to the depth of the deepest beam framing into that joint. Beams and columns modeled using beam elements connect to the panel zone element at the mid-points of the edges of the two panels. These connection points are henceforth referred to as attachment points. There are six attachment points,  $a$  through  $f$ . Attachment points  $a$ ,  $b$ ,  $c$ , and  $d$  are reserved for beams, while attachment points  $e$  and  $f$  are reserved for columns.

The panels are assumed to deform only in shear as a result of the end moments and shears

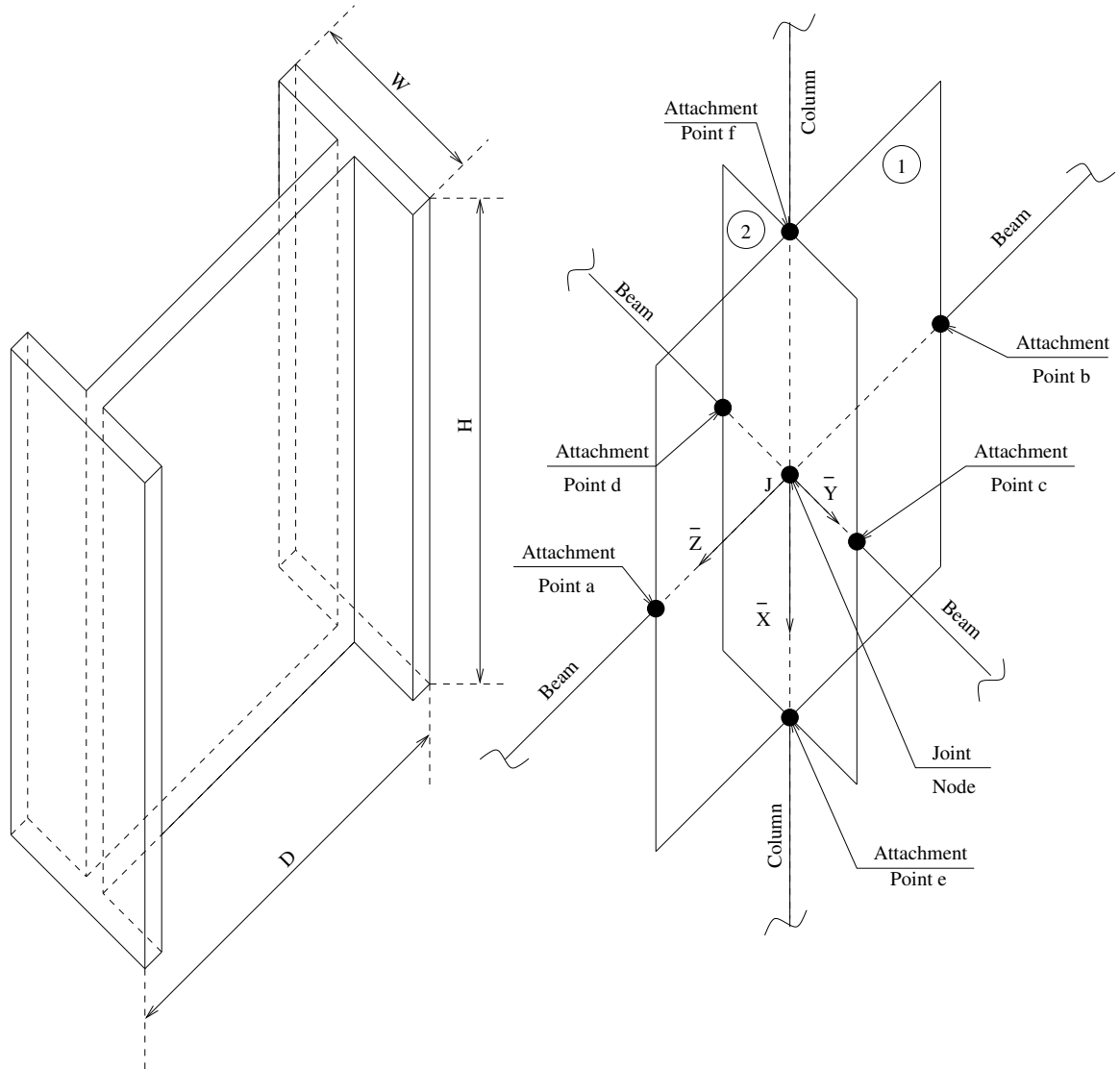


Figure 3.1: Panel Zone Element

of the attached beams and columns. However, they remain planar and perpendicular to each other at all times. The relation between the shear stress,  $\tau$ , and the shear strain,  $\gamma$ , can be a nonlinear relation. Detailed experimental [7, 9, 17, 1] and analytical [16, 8, 7, 17, 4] studies have been carried out on steel-frame joints and have resulted in a much better understanding of their hysteretic behavior. Two material shear stress-strain models (bilinear and linear-quadratic [2, 5]) are implemented in this work.

Each panel has two degrees of freedom as shown in Figure 2.3:  $\theta_{J\bar{Y}}^B$  and  $\theta_{J\bar{Y}}^C$  for panel ①, and  $\theta_{J\bar{Z}}^B$  and  $\theta_{J\bar{Z}}^C$  for panel ②, where  $J$  is the global node at the center of the joint. These DOF are also global DOF. Strain or rotation in one of the panels causes a rigid body rotation but no strain in the orthogonal panel.

The joint coordinate system with origin at the center of the joint (at the node),  $\bar{X}\bar{Y}\bar{Z}$  is defined as follows:

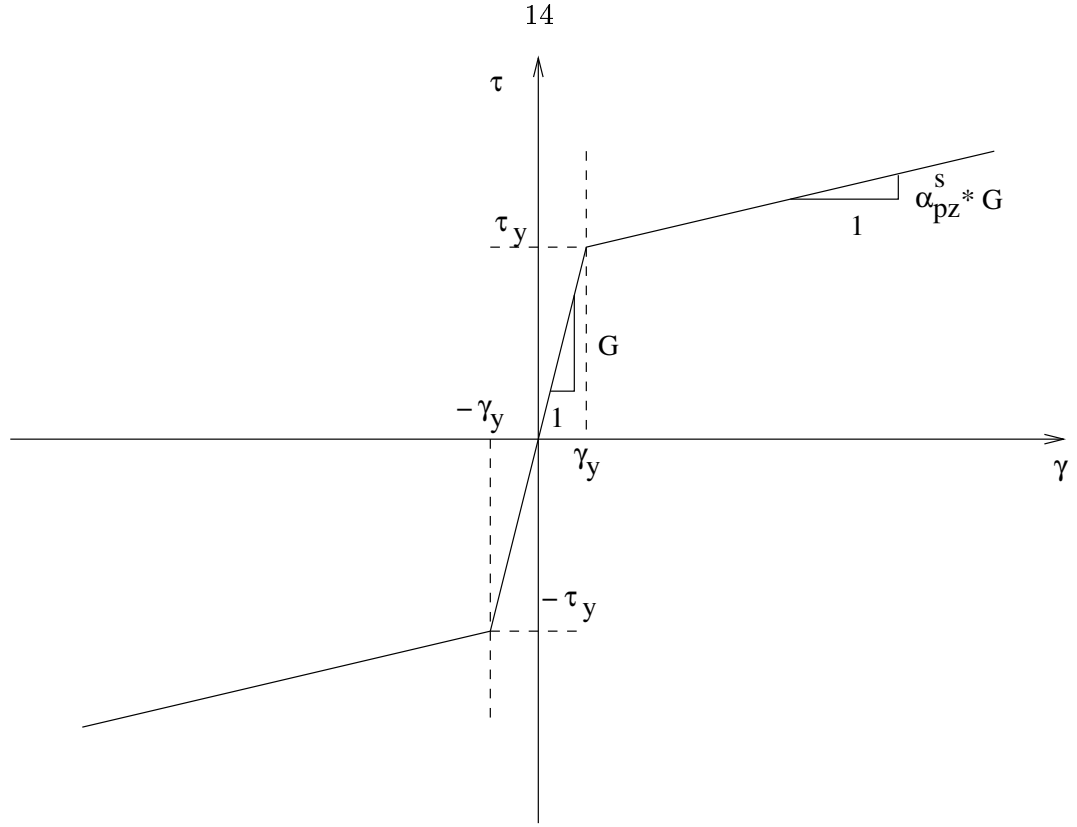
1.  $\bar{X}$  axis: Defined as the vector from attachment point  $f$  to attachment point  $e$  of the joint.
2.  $\bar{Y}$  axis: It is a vector normal to the plane consisting of attachment points  $a$ ,  $b$ ,  $e$  and  $f$ . A unit vector in the direction of  $\bar{Y}$  axis is computed as a cross-product of the vector going from point  $b$  to point  $a$  with the vector going from point  $f$  to point  $e$  and dividing by the length of the resulting vector.
3.  $\bar{Z}$  axis: It is a vector normal to the plane consisting of attachment points  $c$ ,  $d$ ,  $e$ , and  $f$ . A unit vector in the direction of  $\bar{Y}$  axis is computed as a cross-product of the vector going from point  $d$  to point  $c$  with the vector going from point  $f$  to point  $e$  and dividing by the length of the resulting vector.

Thus, at the beginning of the analysis, the joint coordinate system is the same as the local element coordinate system of the associated column of the joint (ACJ).

### 3.1.1 Material Models for Panel Shear

#### Bilinear Model

The bilinear model was first proposed by Fielding and Chen [4]. The virgin shear stress-strain curve for the joint hysteresis behavior in this model is bilinear (Figure 3.2). The



Bilinear shear stress–strain law for a panel zone

Figure 3.2: Shear Stress-Strain Model for a Bilinear Panel Zone Element

virgin curves for positive and negative shears are identical. The shear yield stress,  $\tau_y$ , is given by the von Mises yield criterion:

$$\tau_y = \frac{\sigma_y}{\sqrt{3}} \quad (3.1)$$

where  $\sigma_y$  is the axial yield stress of the material. The yield strain is given by

$$\gamma_y = \frac{\tau_y}{G} \quad \text{where} \quad G = \frac{E}{2(1 + \nu)} \quad (3.2)$$

In the above,  $E$  is the Young's modulus,  $G$  is the shear modulus, and  $\nu$  is the Poisson's ratio of the material. For a panel zone with the bilinear material model, the tangent shear

modulus is given by:

$$G_T = G \text{ if panel zone has not yielded} \quad (3.3a)$$

$$G_T = \alpha_{pz}^s G \text{ if panel zone has yielded} \quad (3.3b)$$

The ratio,  $\alpha_{pz}^s$ , of the post-yield stiffness to the shear modulus modulus, is user-defined. Unloading occurs linearly until zero stress is reached at a rate equal to the elastic modulus and further loading in the reverse direction follows a bilinear stress-strain relation in the opposite direction shifted suitably.

### Linear-Quadratic Ellipsoidal Model

The linear-quadratic ellipsoidal joint hysteresis model proposed by Challa and Hall [2, 5] defines a virgin curve for the joint hysteresis behavior as shown in Figure 3.3. The shear stress-strain is assumed to be linear until a stress of  $0.8\tau_y$  is reached corresponding to a strain of  $0.8\gamma_y$ , where, as before,  $\tau_y$  and  $\gamma_y$  are the yield shear stress and strain respectively. The ultimate shear stress,  $\tau_u$ , of the panel zone is assumed to be equal to  $2.35\tau_y$ , corresponding to an ultimate shear strain,  $\gamma_u$ , of  $100\gamma_y$ . The post-ultimate behavior is assumed to be perfectly plastic. The curve between the joint shear strain limits of  $0.8\gamma_y$  and  $100\gamma_y$  is defined by a quadratic ellipse given by:

$$\frac{(\gamma - \gamma_0)^2}{a^2} + \frac{(\tau - \tau_0)^2}{b^2} = 1 \quad (3.4)$$

where  $\sigma_0$  and  $\gamma_0$  are the coordinates of the center of the ellipse whose semi-major and semi-minor axes are  $a$  and  $b$ , respectively. The hysteresis loops or the branch curves are defined by cubic ellipses given by:

$$-\frac{(\gamma - \gamma_0)^3}{a^3} + \frac{(\tau - \tau_0)^3}{b^3} = 1 \text{ for positive curve} \quad (3.5a)$$

$$\text{and } \frac{(\gamma - \gamma_0)^3}{a^3} - \frac{(\tau - \tau_0)^3}{b^3} = 1 \text{ for negative curve} \quad (3.5b)$$

The hysteresis rules to define the cyclic response of each panel are given in Reference [2]. For a panel zone with the linear-quadratic ellipsoidal material model, the tangent shear modulus,  $G_T$ , is given by the slope of the curve (Figure 3.3) at the current state which is



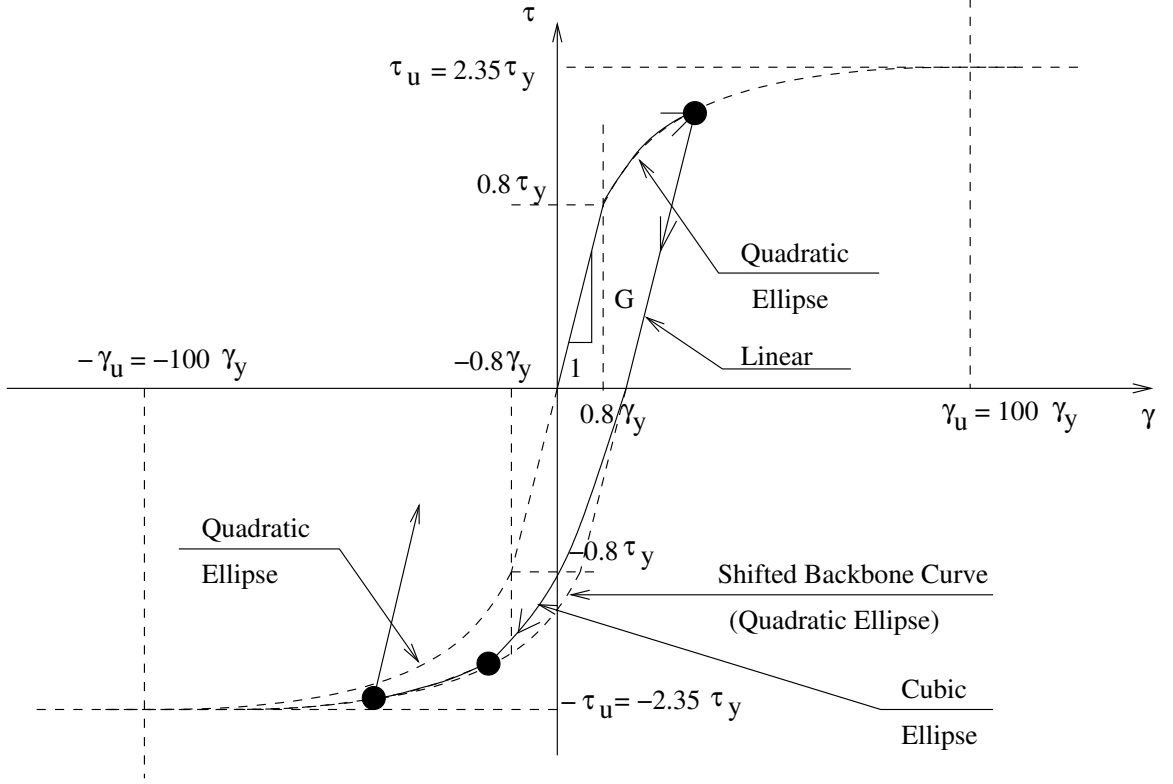


Figure 3.3: Shear Stress-Strain Model for a Linear-Quadratic Panel Zone Element

determined from the stress-strain history and current strain increment.

### 3.2 Plastic Hinge Beam Element

The following assumptions are made in the formulation of this element:

1. The cross-section is assumed uniform along the length of the element.
2. The cross-section is assumed to be doubly symmetric (with shear center at the centroid).
3. Plane sections remain plane; however, they do not have to remain normal to the beam axis. Thus, shear deformations are included.
4. Strains in the element are small.
5. Lateral deflections relative to the chord are small. This means that the effect of bowing [14, 15, 6] on axial stiffness is neglected. This effect is important only for very

slender members or during post-buckling of compression members such as braces in braced frames.

6. Warping restraint under twisting is neglected.
7. The element is loaded only at its ends.
8. Composite action with the slab is not included.
9. Local buckling is not included.

The plastic hinge beam element has two nodes which connect to the attachment points  $a$  through  $f$  of the panel zone element. This element can model beams and columns in framed structures. Columns connect to attachment points  $e$  and  $f$  while beams connect to attachment points  $a$  through  $d$ . The beam element along with its degrees of freedom is shown in Figure 3.4. Original length of the element is  $L_0$ .

The plastic hinge element local coordinate system,  $X'Y'Z'$ , is a right-handed, orthogonal system and is defined as follows:

1.  $X'$  axis: It runs along the longitudinal axis of the element at the centroid of the cross-section. It is defined as a vector from node 1 to node 2 which are also located at the centroid.
2.  $Y'$  axis:  $Y'$  axis is orthogonal to  $X'$  and is the major principal axis of the cross-section of the element.
3.  $Z'$  axis:  $Z'$  axis is the minor principal axis of the cross-section of the element.

The major and minor principal axes,  $Y'$  and  $Z'$ , of the cross-section are oriented using a user-defined orientation angle,  $\alpha_{or}$ , that is defined differently for beams and columns as described below.

For columns,  $\alpha_{or}$  is the angle made by the  $Z'$  axis with a plane passing through the longitudinal axis of the element,  $X'$  and the global  $X$  axis (See Figure 4.1). The direction of  $\alpha_{or}$  is defined using the right-hand-thumb rule and is positive if it is in the positive direction of  $X'$  axis. It is updated after every iteration during each time step of the analysis based on the twist (torsional deformation) in the element.

For beams,  $\alpha_{or}$  is the angle made by the  $Z'$  axis with a plane passing through the longitudinal axis of the element,  $X'$  and the global  $Z$  axis (See Figure 4.1). When the

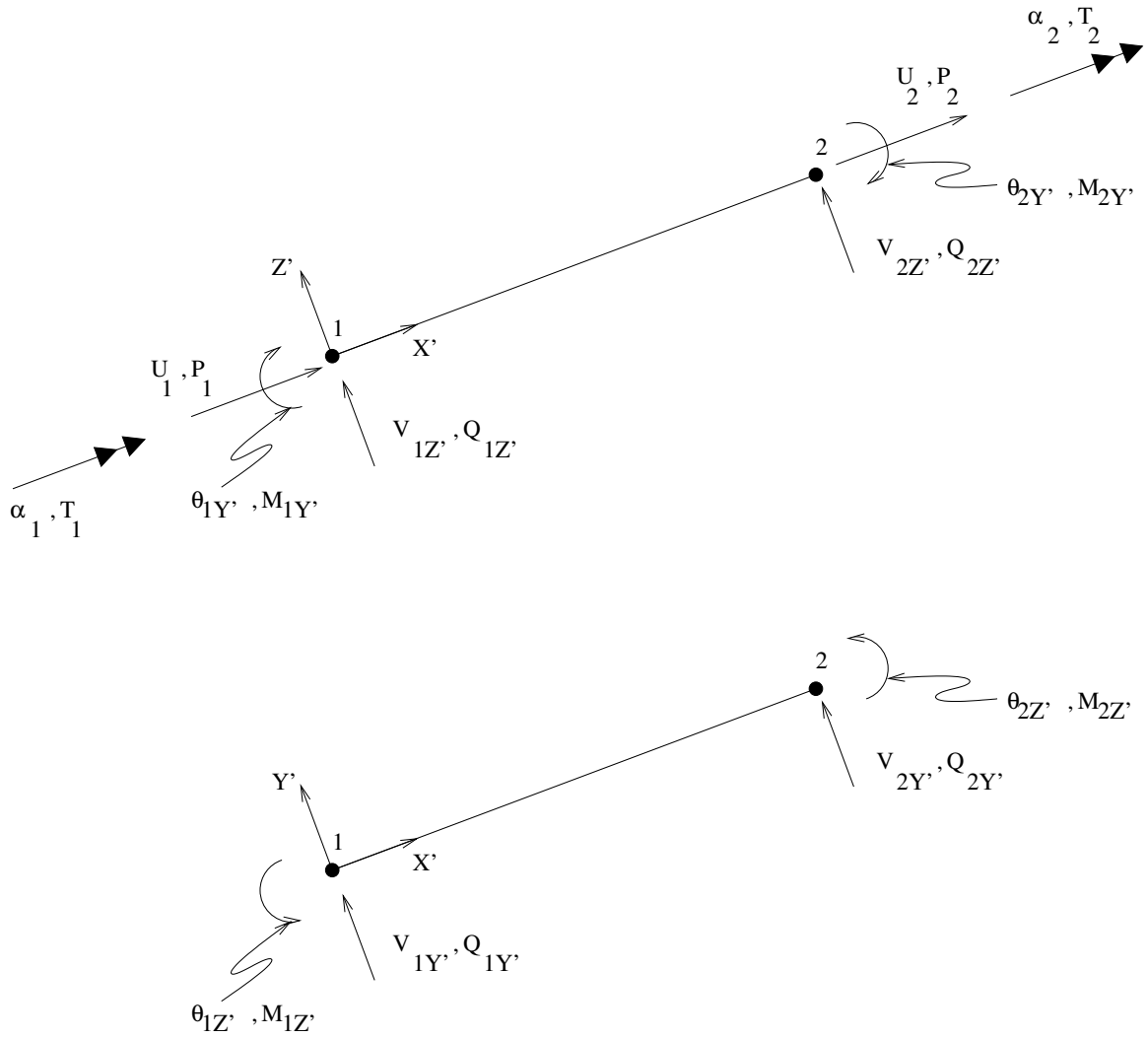


Figure 3.4: DOF of the Plastic Hinge Beam Element Showing Nodal Translations and Rotations and Nodal Forces and Moments

global  $Z$  axis is the vertical axis, this plane is a global vertical plane passing through the longitudinal axis of the element. The direction of  $\alpha_{or}$  is defined using the right-hand-thumb rule and is positive if it is in the positive direction of  $X'$  axis. It is updated after every iteration during each time step of the analysis based on the twist (torsional deformation) in the element.

### 3.2.1 Degrees of Freedom, and Nodal Forces and Moments

The degrees of freedom (Figure 3.4) of the plastic hinge element are:

1.  $U_1, U_2 = X'$  translations at nodes 1 and 2 respectively.
2.  $V_{1Y'}, V_{2Y'} = Y'$  translations at nodes 1 and 2 respectively.
3.  $V_{1Z'}, V_{2Z'} = Z'$  translations at nodes 1 and 2 respectively.
4.  $\alpha_1, \alpha_2 =$  Rotations of the element about  $X'$  at nodes 1 and 2 respectively.
5.  $\theta_{1Y'}, \theta_{2Y'} =$  Rotations of the element about  $Y'$  at nodes 1 and 2 respectively.
6.  $\theta_{1Z'}, \theta_{2Z'} =$  Rotations of the element about  $Z'$  at nodes 1 and 2 respectively.

Corresponding to these DOF are nodal forces and moments (Figure 3.4):

1.  $P_1, P_2 =$  Forces in  $X'$  direction at nodes 1 and 2 respectively.
2.  $Q_{1Y'}, Q_{2Y'} =$  Forces in  $Y'$  direction at nodes 1 and 2 respectively.
3.  $Q_{1Z'}, Q_{2Z'} =$  Forces in  $Z'$  direction at nodes 1 and 2 respectively.
4.  $T_1, T_2 =$  Moments about  $X'$  axis at nodes 1 and 2 respectively.
5.  $M_{1Y'}, M_{2Y'} =$  Moments about  $Y'$  axis at nodes 1 and 2 respectively.
6.  $M_{1Z'}, M_{2Z'} =$  Moments about  $Z'$  axis at nodes 1 and 2 respectively.

### 3.2.2 Internal Forces and Moments

The set of internal forces and moments in the plastic hinge beam element are as follows:

1.  $P =$  Axial force in the element.

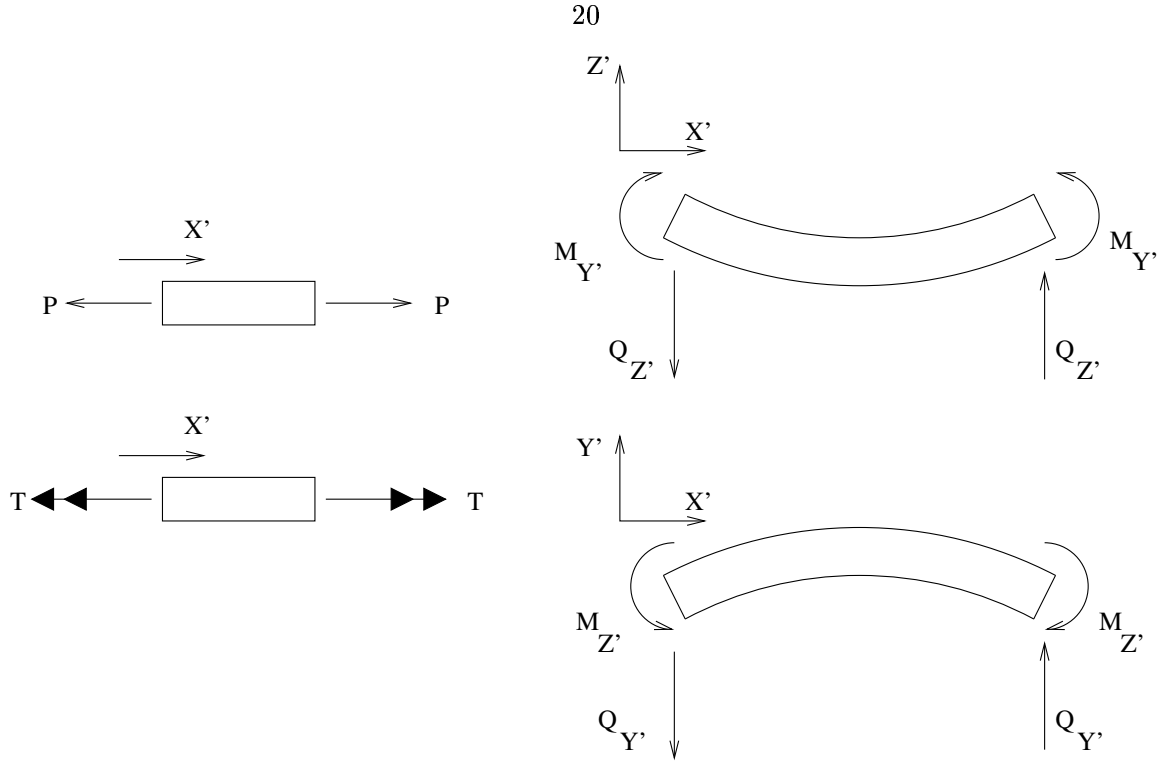


Figure 3.5: Sign Convention for Internal Forces and Couples in a Plastic Hinge Beam Element

2.  $Q_{Y'}$ ,  $Q_{Z'}$  = Shear forces in the element in the  $Y'$  and  $Z'$  directions respectively.
3.  $T$  = Twisting moment in the element.
4.  $M_{Y'}$ ,  $M_{Z'}$  = Moments in the element about the  $Y'$  and  $Z'$  axes respectively.

The sign convention (positive directions) for these forces and moments is shown in Figure 3.5.

### 3.2.3 Material Nonlinearity

Two types of material nonlinearity are considered.

1. Axial yielding when axial force,  $P$ , in the element reaches the yield axial force,  $P_y$ , which is given by:

$$P_y = \sigma_y A \quad (3.6)$$

where  $\sigma_y$  is the yield stress of the material and  $A$  is the area of the cross-section.

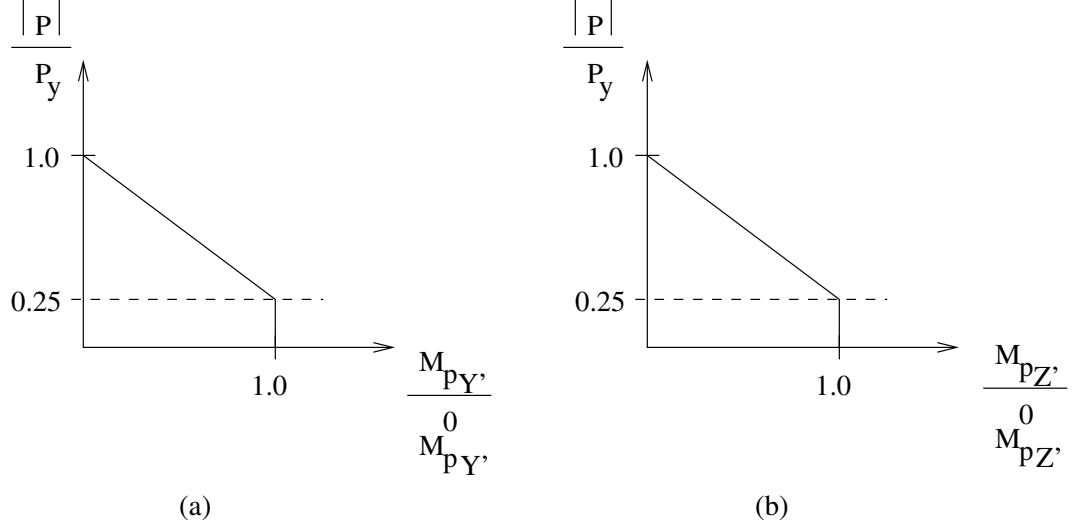


Figure 3.6:  $P - M_{pY'} - M_{pZ'}$  Interaction Diagrams used for the Plastic Hinge Element

2. Flexural yielding when the bending moment,  $M_{Y'}$  or  $M_{Z'}$  at the ends of the element reaches the plastic moment capacity,  $M_{pY'}$  or  $M_{pZ'}$ , which depend on the axial force,  $P$ , as shown in Figure 3.6. The plastic moment capacities when  $P = 0$ , denoted by  $M_{pY'}^0$  and  $M_{pZ'}^0$ , are given by

$$M_{pY'}^0 = \sigma_y Z_{Y'} \quad (3.7a)$$

$$M_{pZ'}^0 = \sigma_y Z_{Z'} \quad (3.7b)$$

where  $Z_{Y'}$  and  $Z_{Z'}$  are the plastic moduli of the cross-section of the element about its major and minor axes respectively. Once  $|M_{Y'}|$  or  $|M_{Z'}|$  reaches  $|M_{pY'}|$  or  $|M_{pZ'}|$  at node 1 or node 2, further loading causes a kink, termed a plastic hinge to form in the beam at that node. Between these plastic hinge locations, the beam behaves elastically. To approximate the effect of strain hardening, elastic rotational springs are mounted across the plastic hinges to exert moments proportional to the kink angles about  $Y'$  and  $Z'$  axes.

### 3.3 Elastofiber Beam Element

The following assumptions are made in the formulation of this element:

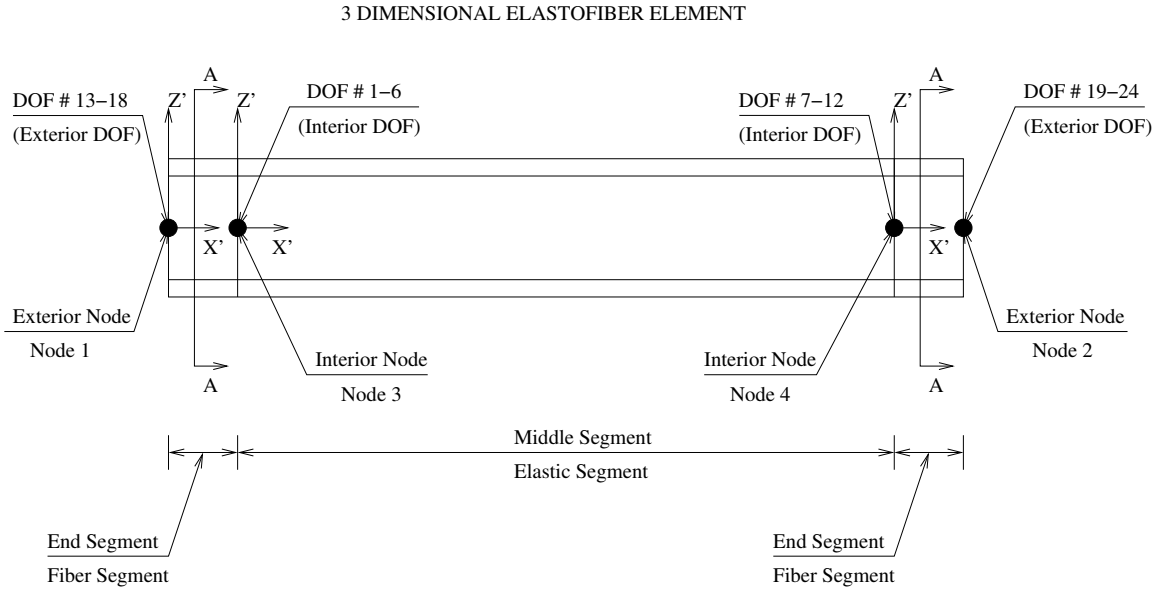
1. The cross-section is assumed uniform along the length of the element.

2. The cross-section is assumed to be doubly symmetric (with shear center at the centroid).
3. Plane sections remain plane; however, they do not have to remain normal to the beam axis. Thus, shear deformations are included.
4. Strains in the element are small.
5. Lateral deflections relative to the chord are small. This means that the effect of bowing [14, 15, 6] on axial stiffness is neglected. This effect is important only for very slender members or during post-buckling of compression members such as braces in braced frames.
6. Warping restraint under twisting is neglected.
7. The element is loaded only at its ends.
8. Composite action with the slab is not included.
9. Local buckling is not included.

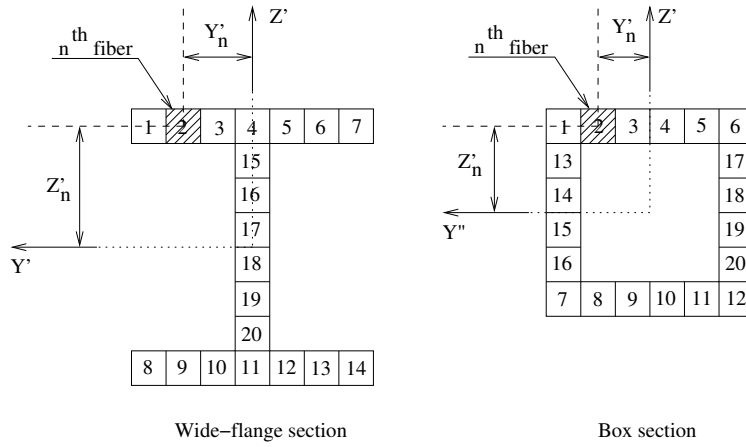
The elastofiber beam element has 3 segments and 4 nodes. Two nodes numbered 1 and 2 are located at the ends and connect to the attachment points  $a$  through  $f$  of the panel zone element. The other two nodes are interior and are numbered 3 and 4 (See Figure 3.7). This element can model beams and columns in framed structures. As in the case of the plastic hinge element, columns connect to attachment points  $e$  and/or  $f$  while beams connect to attachment points  $a$  through  $d$ . Original length of a segment is denoted by  $L_{s0}$ . Segment 1 goes from node 1 to node 3, segment 2 from node 3 to node 4, and segment 3 from node 4 to node 2.

Each segment has its own local coordinate system,  $X'Y'Z'$ , which is a right-handed, orthogonal system and is defined as follows:

1.  $X'$  axis: It runs along the longitudinal axis of the segment at the centroid of the cross-section. It is defined as a vector from node 1 to node 3 for segment 1, from node 3 to node 4 for segment 2, and from node 4 to node 2 for segment 3. All the nodes are located at the centroid of the cross-section.



Note:  $X'Y'Z'$  refers to the segment local coordinate system



Section AA

Figure 3.7: Layout of the 3-Segment Elastofiber Element



2.  $Y'$  axis:  $Y'$  axis is orthogonal to  $X'$  and is the major principal axis of the cross-section of the element.
3.  $Z'$  axis:  $Z'$  axis is the minor principal axis of the cross-section of the element.

Initially, beam element is straight and each of the three  $X'Y'Z'$  systems has the same orientation.  $Y'$  and  $Z'$  axes are oriented in the same way as for the plastic hinge element using the orientation angle,  $\alpha_{or}$ , defined differently for beams and columns as described in Section 3.2. The matrix of direction cosines,  $[C']$ , is computed as described in those sections for segments in beams and columns.

### 3.3.1 Degrees of Freedom of a Segment, and Nodal Forces and Moments

The degrees of freedom (Figure 3.8) of a segment of the elastofiber element are:

1.  $U_i, U_j = X'$  translations at nodes  $i$  and  $j$  of the segment respectively.
2.  $V_{iY'}, V_{jY'} = Y'$  translations at nodes  $i$  and  $j$  respectively.
3.  $V_{iZ'}, V_{jZ'} = Z'$  translations at nodes  $i$  and  $j$  respectively.
4.  $\alpha_i, \alpha_j =$  Rotations of the segment about  $X'$  at nodes  $i$  and  $j$  respectively.
5.  $\theta_{iY'}, \theta_{jY'} =$  Rotations of the segment about  $Y'$  at nodes  $i$  and  $j$  respectively.
6.  $\theta_{iZ'}, \theta_{jZ'} =$  Rotations of the segment about  $Z'$  at nodes  $i$  and  $j$  respectively.

Corresponding to these DOF are nodal forces and moments (Figure 3.8):

1.  $P_i, P_j =$  Forces in  $X'$  direction at nodes  $i$  and  $j$  respectively.
2.  $Q_{iY'}, Q_{jY'} =$  Forces in  $Y'$  direction at nodes  $i$  and  $j$  respectively.
3.  $Q_{iZ'}, Q_{jZ'} =$  Forces in  $Z'$  direction at nodes  $i$  and  $j$  respectively.
4.  $T_i, T_j =$  Moments about  $X'$  axis at nodes  $i$  and  $j$  respectively.
5.  $M_{iY'}, M_{jY'} =$  Moments about  $Y'$  axis at nodes  $i$  and  $j$  respectively.
6.  $M_{iZ'}, M_{jZ'} =$  Moments about  $Z'$  axis at nodes  $i$  and  $j$  respectively.

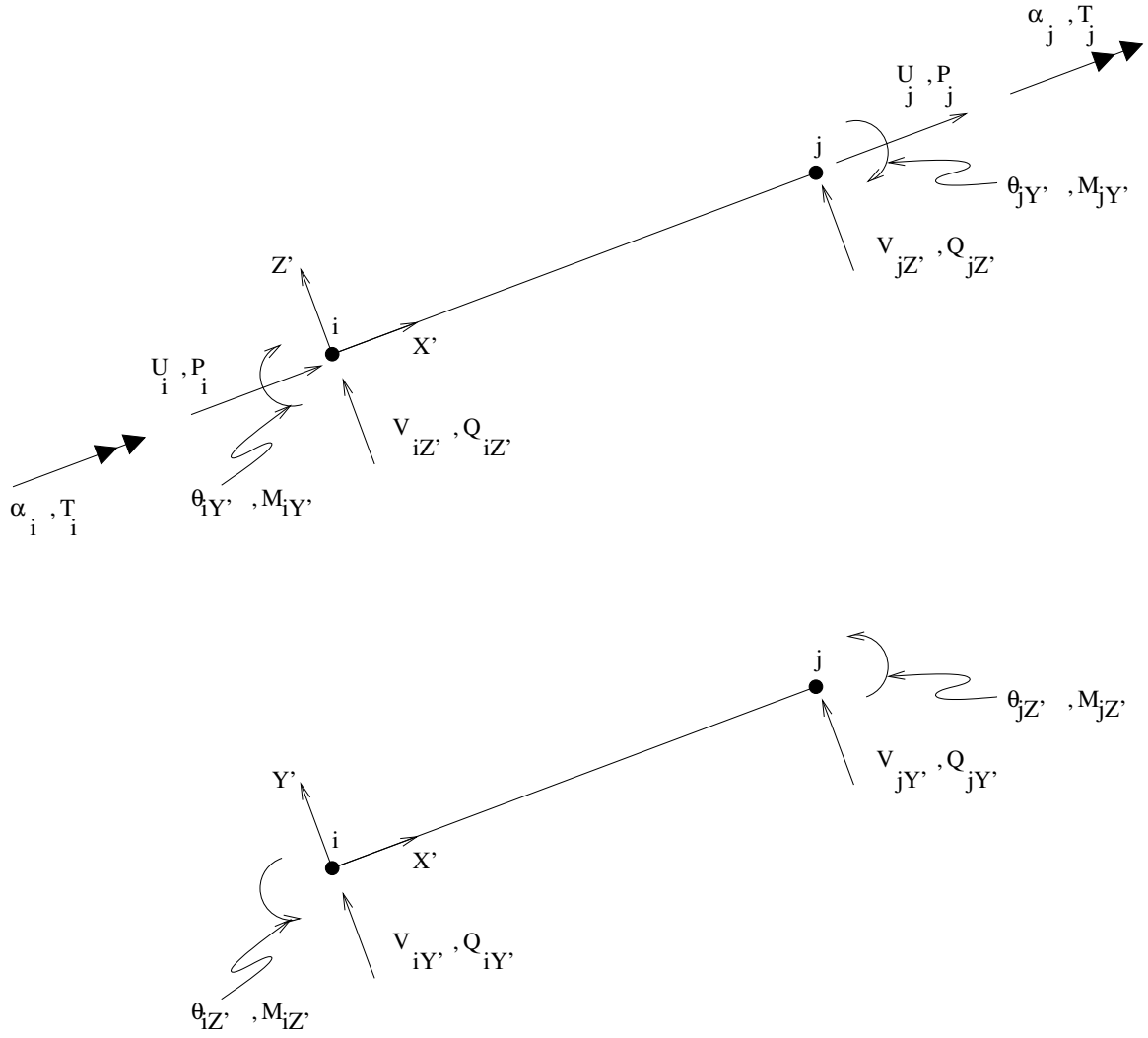


Figure 3.8: DOF of a Segment of the Elastofiber Beam Element Showing Nodal Translations and Rotations and Nodal Forces and Moments

### 3.3.2 Internal Forces and Moments in a Segment

The set of internal forces and moments in a segment of the elastofiber element are as follows:

1.  $P$  = Axial force in the segment.
2.  $Q_{Y'}$ ,  $Q_{Z'}$  = Shear forces in the segment in the  $Y'$  and  $Z'$  directions respectively.
3.  $T$  = Twisting moment in the segment.
4.  $M_{Y'}$ ,  $M_{Z'}$  = Moments in the segment about the  $Y'$  and  $Z'$  axes respectively.

The sign convention (positive directions) for these forces and moments is shown in Figure 3.5.

The middle segment of the elastofiber beam element is an elastic version of the plastic hinge beam element, i.e., no axial yielding and no plastic hinging. End segments are fiber segments. As shown in Figure 3.7, fiber segment cross-sections are divided into  $N$  fibers; each fiber runs the length of the segment. Associated with each fiber is a nonlinear hysteretic stress-strain law for axial stress,  $\sigma_n$ , and axial strain,  $\epsilon_n$ , where  $n$  denotes the  $n^{th}$  fiber. This accounts for nonlinear coupling between bending about the  $Y'$  (major) and  $Z'$  (minor) axes, and axial deformation.

The fiber segment is based on finite element methodology whereby the beam translations and rotations are interpolated linearly and independently from the nodal values. This requires a one-point integration on the shear terms to prevent locking.

Because of the presence of the interior nodes, the updating process for each elastofiber beam element requires a nonlinear structural analysis. Iterations for each element are performed within each global iteration. The multi-segment element analyses are performed with degrees of freedom transformed to the global coordinate system,  $XYZ$ .

## 3.4 Material Model for Fiber Axial Stress-Strain

The axial stress-strain behavior of the fiber is governed by the hysteretic uniaxial cubic-ellipsoidal law proposed by Challa [2, 5]. The stress-strain curve for monotonic loading, shown in Figure 3.9, is symmetric about both the strain and stress axes. It consists of an elastic portion, a yield plateau and a strain-hardening region which is described by a cubic ellipse. The monotonic stress-strain curve is characterized by seven parameters: the

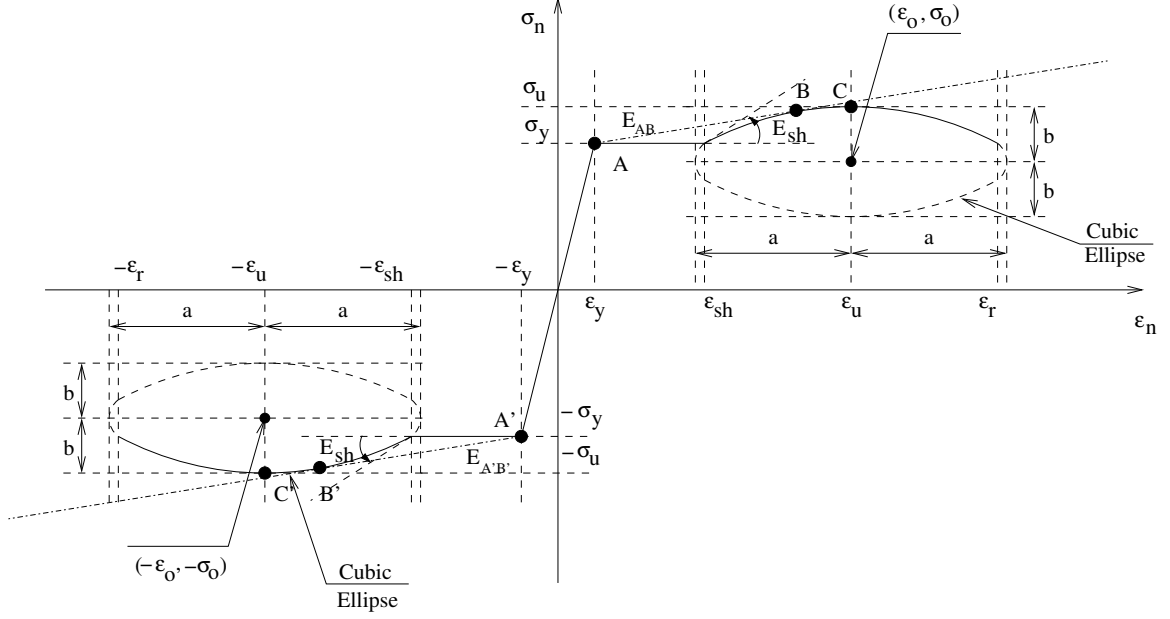


Figure 3.9: Stress-Strain Model for a Typical Fiber in a Fiber Segment of an Elastofiber Element

yield stress,  $\sigma_y$ , the ultimate stress,  $\sigma_u$ , the yield strain,  $\epsilon_y$ , the strain at initiation of strain hardening,  $\epsilon_{sh}$ , the strain at ultimate stress,  $\epsilon_u$ , the rupture strain,  $\epsilon_r$ , and the tangent modulus at the initiation of strain hardening,  $E_{sh}$ . The cubic ellipse describing the strain hardening region is assumed to be centered at  $(\epsilon_0 = \epsilon_u, \sigma_0)$  with radii,  $a$  and  $b$ :

$$\frac{(\epsilon_n - \epsilon_0)^3}{a^3} + \frac{(\sigma_n - \sigma_0)^3}{b^3} = 1 \quad (3.8)$$

Parameters,  $\sigma_0$ ,  $a$ , and  $b$ , are determined from the user-defined properties,  $\epsilon_{sh}$ ,  $\epsilon_u$ ,  $E_{sh}$ ,  $\sigma_y$ , and  $\sigma_u$ .

The hysteresis laws required to describe the stress-strain behavior under cyclic loading or reversal of loading are described in [2] and [5].

### 3.5 Elastic 4-Noded Diaphragm Element

Floor slabs in buildings act as diaphragms that tie the moment frames together such that they act as one unit. In FRAME3D, the floor slabs are modeled using 4-noded elastic diaphragm elements. The diaphragm element is a 4-noded linear-elastic plane element that can take a quadrilateral form in 3D spatial coordinates,  $XYZ$ . It is used to provide trans-

lational restraint caused by floor slabs to columns in buildings. The following assumptions are made in the formulation of this element:

1. Material is isotropic and linear-elastic, with Young's modulus,  $E$ , shear modulus,  $G$ , and Poisson's ratio,  $\nu$ .
2. Warping is not considered.
3. A state of plane-stress is assumed in the element.
4. The element continues to provide the same restraint (in magnitude and direction) as in the original configuration even though during the course of the analysis the nodes of the element may have translated causing the geometry of the element to change.
5. The element is not externally loaded and all the stresses in the element are generated by stiffness forces due to the translation of the nodes during the course of the analysis.

The nodes of the element are labeled  $i$ ,  $j$ ,  $m$ , and  $n$ . Each element has its own local coordinate system,  $X'Y'$ , which lies in the plane of the element and is defined as follows:

1.  $X'$  is in the direction of a vector from node  $j$  to node  $k$ .
2.  $Y'$  is orthogonal to  $X'$  and lies in the plane of the 4 nodes.

### 3.5.1 Degrees of Freedom, and Nodal Forces

The degrees of freedom (Figure 3.10) of a diaphragm element are:

1.  $V_{iX'}$ ,  $V_{jX'}$ ,  $V_{mX'}$ ,  $V_{nX'}$  =  $X'$  translations at nodes  $i$ ,  $j$ ,  $m$ , and  $n$  of the element respectively.
2.  $V_{iY'}$ ,  $V_{jY'}$ ,  $V_{mY'}$ ,  $V_{nY'}$  =  $Y'$  translations at nodes  $i$ ,  $j$ ,  $m$ , and  $n$  of the element respectively.

Corresponding to these degrees of freedom are nodal forces:

1.  $Q_{iX'}$ ,  $Q_{jX'}$ ,  $Q_{mX'}$ ,  $Q_{nX'}$  =  $X'$  forces at nodes  $i$ ,  $j$ ,  $m$ , and  $n$  of the element respectively.
2.  $Q_{iY'}$ ,  $Q_{jY'}$ ,  $Q_{mY'}$ ,  $Q_{nY'}$  =  $Y'$  forces at nodes  $i$ ,  $j$ ,  $m$ , and  $n$  of the element respectively.

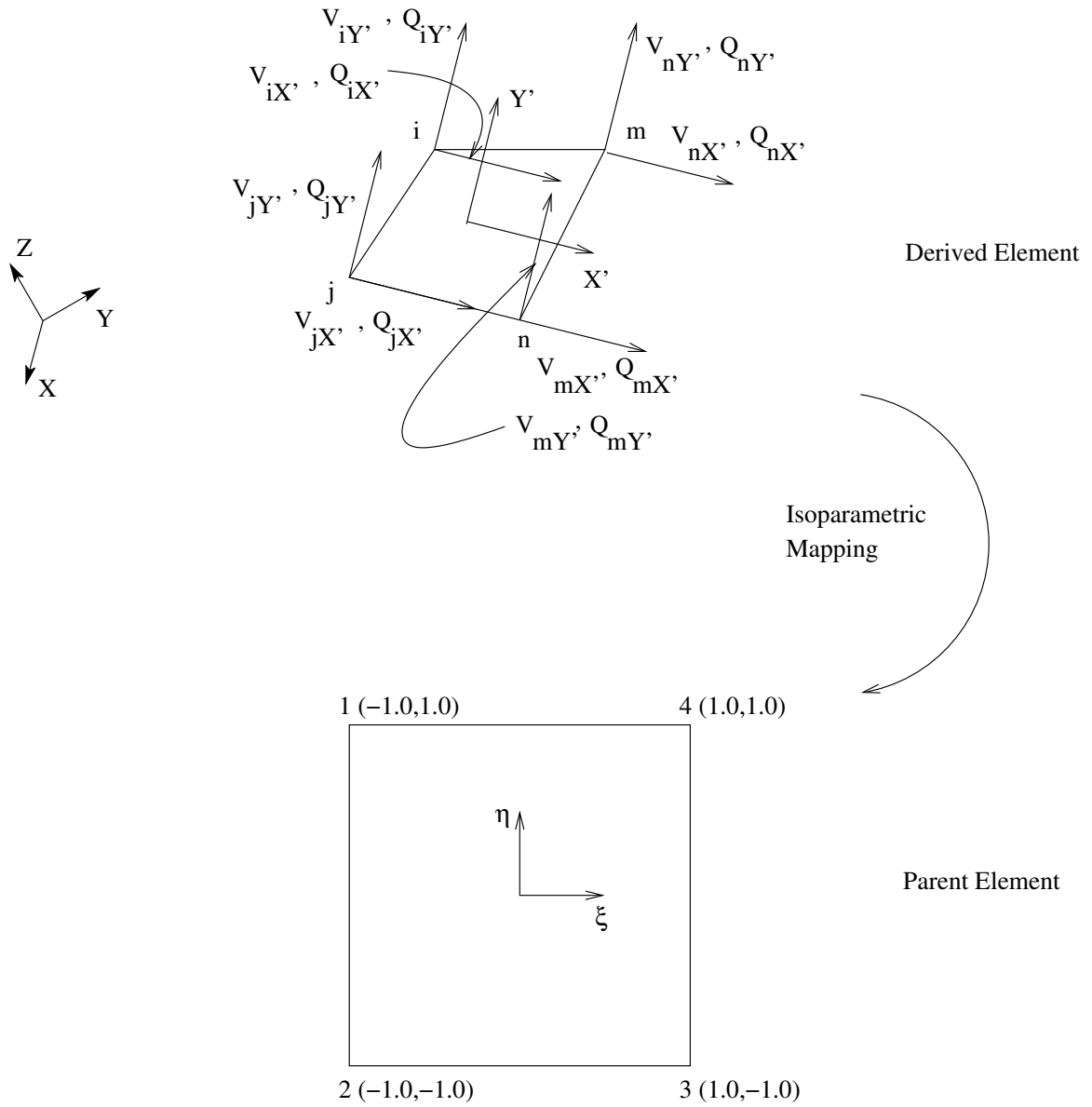


Figure 3.10: DOF of the Diaphragm Element Showing Nodal Translations and Rotations and Nodal Forces and the Isoparametric Mapping from a Square Parent Element

### 3.5.2 Internal Stresses in a Diaphragm Element

The internal stresses in the diaphragm element are:

1.  $\sigma_{X'X'}$  = Normal stress in  $X'$  direction.
2.  $\sigma_{Y'Y'}$  = Normal stress in  $Y'$  direction.
3.  $\sigma_{X'Y'}$  = Shear stress in the element.

These can be transformed to principal tensile and compressive stresses,  $\sigma_1$  and  $\sigma_2$  respectively, using Mohr's circle.

## 3.6 Elastic Translational or Rotational Spring Element

The program includes elastic translational and rotational spring elements. Springs can be used to model foundations. They can also be used to retrieve reaction forces and moments at the supports. Springs must be oriented in one of the three directions of the global coordinate system,  $XYZ$ .

## Chapter 4 Input Data

Five input files are required. These are listed in Table 4.1.

Table 4.1: Input Files for FRAME3D Analysis

File Name	Description
<i>for005</i>	Structural model input file.
<i>for009</i>	Section property database.
<i>for090</i>	Global $X$ direction acceleration record.
<i>for091</i>	Global $Y$ direction acceleration record.
<i>for092</i>	Global $Z$ direction acceleration record.

### 4.1 Input File for005

Structure input should be prepared in a file named “for005”. All input data can be prepared in free format. There are 17 data blocks each separated by atleast one comment/blank line (the actual number of comment/blank lines are given in Table 4.2). Note that the comment/blank lines are always required even if there’s no data required in a certain data block.

The following subsections detail the data format for the input file, *for005*. Each subsection is divided into three parts:

1. Subsubsection, “Format”, gives the sequence of entries to be provided in each data line of the file. Each entry corresponds to a single variable.
2. Subsubsection, “Description”, gives a tabular description in the form:

#### Variable Field Note Definition

**Variable** is the name of the variable/property.

**Field** is the locator of the **Variable** in the current data line. Thus, if the current **Variable** is the third number on the current data line, then, **Field** would be 3.

**Note** is the reference to the note relevant to the current **Variable** in Subsubsection “Notes”.



**Definition** gives a short description of the **Variable**.

3. Subsubsection, “Notes”, gives a list of notes describing the **Variable** in greater detail.

Table 4.2: File for005: List of Data Blocks

S.No.	Data Block	When Needed	No.of Comment/Blank Lines Preceding the Data Block
1.	Heading	Always	1
2.	Main Control Data	Always	1
3.	Integration and Iteration Data	Always	1
4.	Nodal Data	Always	1
5.	Spring Element Data	Only if $NSPR > 0$	1
6.	Beam Element Data	Only if $NPHEL > 0$ or $NFIBEL > 0$	1
7.	Categories for Elastofiber Element Fiber Area Reduction	Always	1
8.	Seed for Random Number Generation for Elastofiber Element Fiber Fracture Stress Computation	Always	1
9.	Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution	Always	1
10.	Panel Zone Element Data	Only if $RYRES > 0$ or $RZRES > 0$ for atleast one node	3
11.	Diaphragm Element Control Data	Always	1
12.	Diaphragm Element Material Data	Only if $NUMPLSTRMAT > 0$	1
13.	Diaphragm Element Connectivity Data	Always	4
14.	Diaphragm Element Output Control Data	Only if $NPLSTRSS > 0$	2
15.	Time History Output Control Data	Only if $NRTH > 0$	3
16.	Ground Acceleration Scaling Factors	Always	1
17.	Deformed Shape Output Control Data	Always	1

#### 4.1.1 Heading

Prepare one line of data upto 72 characters describing the model for output labeling.

#### 4.1.2 Main Control Data

##### Format

Prepare one line of data to define the control parameters of the structural model in the following form:

**NNP NPHEL NFIBEL NPZ0 NPZ1 NSPR MTP NDIM MBD NSSTPS  
NDSTPS NRTH NGITMAX NLITMAX NOUT**

##### Description

Variable	Field	Note	Definition
<i>NNP</i>	1	(1)	Number of node points.
<i>NPHEL</i>	2	(2)	Number of plastic hinge beam elements.
<i>NFIBEL</i>	3	(3)	Number of elastofiber beam elements.
<i>NPZ0</i>	4	(4)	Number of panel zones with bilinear shear stress-strain relation.
<i>NPZ1</i>	5	(5)	Number of panel zones with linear-quadratic shear stress-strain relation.
<i>NSPR</i>	6	(6)	Number of translational and rotational springs.
<i>MTP</i>	7	(7)	Maximum number of turning points.
<i>NDIM</i>	8	(8)	Storage parameter for turning point locations.
<i>MBD</i>	9	(9)	Global half bandwidth.
<i>NSSTPS</i>	10	(10)	Number of static analysis steps.
<i>NDSTPS</i>	11	(11)	Number of dynamic analysis steps.
<i>NRTH</i>	12	(12)	Number of response time histories to be output.
<i>NGITMAX</i>	13	(13)	Maximum number of global iterations per time step.
<i>NLITMAX</i>	14	(14)	Maximum number of local member (elastofiber element) iterations per time step.
<i>NOUT</i>	15	(15)	Output data is written for each <i>NOUT<sup>th</sup></i> time step.

##### Notes

1. This variable defines the number of nodal points required by the user to describe the structural geometry. There can then be a maximum of  $NNP * 8$  degrees of freedom in the model.

2. This variable defines the number of beam elements that are to be modeled as plastic hinge elements with bilinear moment-rotation relation. Identify these elements in the beam element data block by setting the variable, *IELTYPE*, to 0.
3. This variable defines the number of beam elements that are to be modeled as elastofiber elements with cubic-ellipsoidal axial stress-strain relation for each fiber. Identify these elements in the beam element data block by setting the variable, *IELTYPE*, to 1.
4. This variable defines the number of panel-zones that are to be modeled using a bilinear material shear stress-strain relation. Identify these panel zones in the panel zone element data block by setting the variable, *IPZTYPE*, to 0. Note that each panel zone element can have 2 mutually perpendicular panel zones. Data for each of these panel zones needs to be provided separately.
5. This variable defines the number of panel-zones that are to be modeled using a linear-quadratic material shear stress-strain relation. Identify these panel zones in the panel zone element data block by setting the variable, *IPZTYPE*, to 1. Note that each panel zone element can have 2 mutually perpendicular panel zones. Data for each of these panel zones needs to be provided separately.
6. This variable defines the number of elastic springs to be specified in the model. Springs can be used to model foundations. They can also be used to retrieve reaction forces and moments at the supports. In this case, the spring constant/stiffness (the variable, *SPRSTF*, in the Spring Element data block) should be specified to be much greater (atleast four orders of magnitude greater) than the stiffness of the other elements in the model.
7. This variable is used to set the storage for coordinates of the turning points in the stress-strain history of each fiber of the elastofiber element. A reasonable value would be between 20 and 40.
8. This variable defines the dimension of the array storing turning point data for all fibers of all elastofiber elements in the model. This variable need not exceed  $NFIBEL * 2 * 20 * MTP$ . It can be far less than this because most fibers in the structural model would not traverse through *MTP* turning points in their stress-strain history.

9. This variable is used to set the storage of the stiffness and damping matrices. These matrices are stored in banded form and the user needs to specify the maximum global half-bandwidth. The program itself computes a half-bandwidth based on the structural geometry and degrees of freedom. If the user-specified *MBD* is less than that computed internally by the program, then program is terminated with a warning printed in the output file, *for006*.
10. This variable defines the number of steps in which the user-specified static loads in the Nodal data block are to be applied. These loads are divided by this number and this fraction is applied incrementally at each of the static load steps.
11. This variable defines the number of steps for which dynamic loads are to be applied to the structure. Dynamic loads can be specified by providing time-histories of three mutually orthogonal components of acceleration in input files *for090*, *for091*, and *for092*, that accelerate the nodal masses specified in the Nodal data block inducing dynamic inertial forces in the structure. *NDSTPS* can be less than the number of entries in the acceleration records.
12. This variable defines the number of response time histories for which output data is required. The time-history data will be written out in two separate files *for006* and *RTH*. Response quantities include nodal displacements, rotations, spring forces, beam element forces, moments, plastic rotations, axial strain, joint moments, shear strains, and elastofiber element fiber stresses and strains. *NRTH* is limited to 200 due to output formatting constraints.
13. This variable defines the maximum number of global iterations to be implemented at each time-step for convergence to the correct solution. If convergence is not achieved within *NGITMAX* iterations, a warning is printed out in the output file, *for006*, and analysis proceeds to the next time step. A reasonable value for this variable would be in the range of 100-500.
14. This variable defines the maximum number of local member iterations to be implemented for each elastofiber element at each global iteration of each time-step. If convergence is not achieved within *NLITMAX* iterations, a warning is printed out in the output file, *for006*, and analysis proceeds to the next element in the current

global iteration of the current time-step. A reasonable value for this variable would be in the range of 50-200.

15. This variable controls the frequency of time-history output printing. If data is required to be written out at each time-step, then *NOUT* should be assigned the value, 1. If data is required to be written out only every fourth time-step, then *NOUT* should be assigned the value, 4.

### 4.1.3 Integration and Iteration Data

#### Format

Prepare one line of data to define the control parameters for integration and iteration during the solution process in the following form:

**DT TIMEOSTF BETA GAMMA A0 A1 AGRAV STFFACPHE STFFACFE  
STFFACPZ0 STFFACPZ1 TOLF TOLM TOLFIBELF TOLFIBELM THRESH  
NTHR ITHR**

**Description**

<b>Variable</b>	<b>Field</b>	<b>Notes</b>	<b>Definition</b>
<i>DT</i>	1	(1)	Time increment for dynamic time-history analysis.
<i>TIMEOSTF</i>	2	(2)	Time till which the initial elastic global stiffness matrix is to be used for Newton-Raphson iteration.
<i>BETA</i>	3	(3)	Newmark integration parameter.
<i>GAMMA</i>	4	(3)	Newmark integration parameter.
<i>A0</i>	5	(4)	Rayleigh damping parameter.
<i>A1</i>	6	(4)	Rayleigh damping parameter.
<i>AGRAV</i>	7	(5)	Acceleration due to gravity.
<i>STFFACPHE</i>	8	(6)	Fraction of elastic stiffness of plastic hinge element to be added to its tangent stiffness for Newton-Raphson iteration.
<i>STFFACFE</i>	9	(7)	Fraction of elastic stiffness of elastofiber element to be added to its tangent stiffness for Newton-Raphson iteration.
<i>STFFACPZ0</i>	10	(8)	Fraction of elastic stiffness of bilinear panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
<i>STFFACPZ1</i>	11	(9)	Fraction of elastic stiffness of linear-quadratic panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
<i>TOLF</i>	12	(10)	Force tolerance for global convergence of displacement DOF.
<i>TOLM</i>	13	(11)	Moment tolerance for global convergence of rotation DOF.
<i>TOLFIBELF</i>	14	(12)	Force tolerance for local convergence of displacement DOF of an elastofiber element.
<i>TOLFIBELM</i>	15	(13)	Moment tolerance for local convergence of rotation DOF of an elastofiber element.
<i>THRESH</i>	16	(14)	Threshold value for system picture.
<i>NTHR</i>	17	(14)	Threshold node for system picture.
<i>ITHR</i>	18	(14)	Threshold DOF for system picture.

**Notes**

1. This variable defines the time increment for dynamic loading. The acceleration records specified in files *for090*, *for091*, and *for092*, should consist of data points that are spaced at *DT* time units.
2. This variable defines the time until which the initial elastic global stiffness matrix is to



be used for Newton-Raphson iteration. The Newton-Raphson method requires that the tangent stiffness matrix be generated and factored for equation solving in every iterative cycle. This is expensive. Thus, the user can pick an alternate strategy of using the initial stiffness matrix to iterate for a specified number of time steps and then switching to the tangent matrix formulation. This way, for the initial portion of the ground motion time-history, when the structure is still elastic or has just entered the nonlinear regime, huge computation time savings can be achieved, since the constant initial stiffness matrix needs to be factored only once. Of course, the solution process would require a larger number of iterations for convergence during these time steps but this is more than offset by the savings from matrix-factoring. If convergence with the initial stiffness matrix is too slow, try using the tangent stiffness matrix.

In some cases, the solution using the tangent stiffness matrix may not converge, but could keep cycling back-and-forth between two values. Using the initial stiffness matrix may help to converge to the solution without getting lost, under these circumstances.

3. Newmark's method is unconditionally stable if  $\gamma \geq 0.5$  and  $\beta \geq \frac{(2\gamma+1)^2}{16}$ . Artificial positive damping is introduced if  $\gamma > 0.5$  and artificial negative damping if  $\gamma < 0.5$ . If  $\gamma = 0.5$  and  $\beta = 0$ , Newmark's method reduces to the central difference method. A good choice of parameters which makes this method unconditionally stable in linear problems is  $\gamma = 0.5$  and  $\beta = 0.25$ . In this case, Newmark's method reduces to the constant average acceleration method or the trapezoidal method. With these values for  $\gamma$  and  $\beta$  there are no amplitude errors in any sine-wave motion, regardless of its frequency. But periods of sine-wave motion are overestimated. The error increases as  $\frac{\Delta t}{T}$  increases, where  $T$  is the correct period of the motion. Unconditional stability in linear problems however, is no guarantor for the algorithm to remain unconditionally stable when used to solve nonlinear problems.
4. The program assumes that the structural system exhibits Rayleigh damping, i.e. mass and stiffness proportional. Thus, the damping matrix is computed as

$$[D] = a_0 [M] + a_1 [K] \quad (4.1)$$

where  $a_0$  and  $a_1$  are user defined proportionality constants. The following discussion can be of help to the user in selecting reasonable values for  $a_0$  and  $a_1$ .

In the linear elastic case, where modal superposition is valid, if Rayleigh damping is assumed, the modal damping in the  $i^{th}$  mode can be written in terms of the corresponding modal frequency as

$$\zeta_i = \frac{1}{2} \left( a_0 \omega_i + \frac{a_1}{\omega_i} \right) \quad (4.2)$$

Based on this relation, if we plot  $\omega_i$  vs  $\zeta_i$ , the curve is found to be reasonably flat in the intermediate frequency range. Thus, assuming that the structure exhibits approximately equal damping values in all the modes, two frequencies,  $\omega$  and  $R\omega$ , are picked covering the frequency range of interest and a desired amount of damping. Then  $a_0$  and  $a_1$  can be computed as

$$a_0 = \frac{2\zeta\omega R}{1+R} \quad (4.3a)$$

$$a_1 = \frac{2\zeta}{\omega(1+R)} \quad (4.3b)$$

5. This variable defines the acceleration due to gravity and is used to convert the nodal weights specified in the Nodal data block to nodal masses. If nodal masses are specified in the nodal data block instead of weights, then *AGRAV* should be given a value of 1.0.
6. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all plastic hinge beam elements for iterating in order to keep the stiffness matrix from becoming ill-conditioned. A reasonable value for this would be between 0.01 and 0.02.
7. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all elastofiber beam elements for iterating in order to keep the stiffness matrix from becoming ill-conditioned. A reasonable value for this would be between 0.01 and 0.02.
8. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all bilinear panel zone elements for iterating in order to

keep the stiffness matrix from becoming ill-conditioned. A reasonable value for this would be between 0.01 and 0.02.

9. This variable defines the fraction of the elastic stiffness matrix to be added to the tangent stiffness matrix of all linear-quadratic panel zone elements for iterating in order to keep the stiffness matrix from becoming ill-conditioned. A reasonable value for this would be between 0.01 and 0.02.
10. This variable defines the displacement degree of freedom global convergence tolerance on force. If the unbalanced forces corresponding to all the displacement degrees of freedom are less than  $TOLF$  and the unbalanced moments corresponding to all the rotational degrees of freedom are less than  $TOLM$ , the global iteration is deemed to have converged and analysis proceeds to the next time step.
11. This variable defines the rotational degree of freedom global convergence tolerance on moment. If the unbalanced forces corresponding to all the displacement degrees of freedom are less than  $TOLF$  and the unbalanced moments corresponding to all the rotational degrees of freedom are less than  $TOLM$ , the global iteration is deemed to have converged and analysis proceeds to the next time step.
12. This variable defines the displacement degree of freedom member local iteration convergence tolerance on force (applicable to elastofiber elements only). If the unbalanced forces corresponding to all the displacement degrees of freedom are less than  $TOLFIBELF$  and the unbalanced moments corresponding to all the rotational degrees of freedom are less than  $TOLFIBELM$ , the member local iteration is deemed to have converged and analysis proceeds to the next elastofiber element in the current global iteration of the current time step.
13. This variable defines the rotational degree of freedom member local iteration convergence tolerance on moment (applicable to elastofiber elements only). If the unbalanced forces corresponding to all the displacement degrees of freedom are less than  $TOLFIBELF$  and the unbalanced moments corresponding to all the rotational degrees of freedom are less than  $TOLFIBELM$ , the member local iteration is deemed to have converged and analysis proceeds to the next elastofiber element in the current global iteration of the current time step.

14. The variable, *THRESH*, defines the threshold value of the degree of freedom (1-8), *ITHR*, of node, *NTHR*, beyond which the system picture is printed out in file, *for006*. The system picture consists of element forces, plastic rotations and stresses, and nodal displacements.

#### 4.1.4 Nodal Data

##### Format

Prepare *NNP* lines of data, one for each node, to define the data pertaining to nodes of the structural model in the following form:

**X Y Z XRES YRES ZRES RXRES RYRES RZRES FX FY FZ MX MY  
MZ IDASSCOL**

##### Description

Variable	Field	Notes	Definition
<i>X</i>	1	(1)	X coordinate of node-point.
<i>Y</i>	2	(1)	Y coordinate of node-point.
<i>Z</i>	3	(1)	Z coordinate of node-point.
<i>XRES</i>	4	(2)	Restraint to nodal translation in global <i>X</i> direction.
<i>YRES</i>	5	(2)	Restraint to nodal translation in global <i>Y</i> direction.
<i>ZRES</i>	6	(2)	Restraint to nodal translation in global <i>Z</i> direction.
<i>RXRES</i>	7	(3,7)	Restraint to nodal rotation about joint local $\bar{X}$ axis.
<i>RYRES</i>	8	(4,7)	Restraint to nodal rotation about joint local $\bar{Y}$ axis.
<i>RZRES</i>	9	(4,7)	Restraint to nodal rotation about joint local $\bar{Z}$ axis.
<i>FX</i>	10	(5)	Static nodal force in the global <i>X</i> direction.
<i>FY</i>	11	(5)	Static nodal force in the global <i>Y</i> direction.
<i>FZ</i>	12	(5)	Static nodal force in the global <i>Z</i> direction.
<i>MX</i>	13	(6)	Nodal mass in weight units in the global <i>X</i> direction.
<i>MY</i>	14	(6)	Nodal mass in weight units in the global <i>Y</i> direction.
<i>MZ</i>	15	(6)	Nodal mass in weight units in the global <i>Z</i> direction.
<i>IDASSCOL</i>	16	(7)	Element number of the column that is to be associated with the joint under consideration.

##### Notes

1. The *X*, *Y*, and *Z* coordinates of all the node-points should be specified in the global coordinate system. It is recommended that the global *X* and *Y* axes be oriented in the two principal directions of the building and the global *Z* axis be oriented in the vertically upward direction.
2. If the node is free to translate in the specified direction, this variable should be assigned a value of 1. If its not, then it should be assigned a value of 0. Translational restraints can be specified in only 3 directions, the global *X*, *Y*, and *Z* directions.

3. Rotational restraints need to be specified for the node about the 3 axes of the panel-zone (joint) coordinate system. The panel zone coordinate system starts out to be identical to the element coordinate system corresponding to the associated column of the joint (ACJ). Thus, rotation about the  $\bar{X}$  axis corresponds to the twisting DOF of the joint. If the joint is not free to twist, this variable should be assigned a value of 0. If it is free to rotate, then this variable should be assigned a value of 1.
4. Rotational restraints need to be specified for the node about the 3 axes of the panel-zone (joint) coordinate system. The panel zone coordinate system starts out to be identical to the element coordinate system corresponding to the associated column of the joint (ACJ). Thus, rotations about the  $\bar{Y}$  and  $\bar{Z}$  axes correspond to rotations of the joint about the ACJ major and minor axes respectively. If the joint is not free to rotate, this variable should be assigned a value of 0. If it is free to rotate and if panel zone effects in the direction under consideration are not required to be considered, then this variable should be assigned a value of 1. If panel zone effects are to be considered, then this variable should be assigned a value of 2.
5. These variables define the static forces that are to be applied to the current node in the global  $X$ ,  $Y$ , and  $Z$  directions. These static forces can be applied in *NSSTPS* steps (a fraction of these loads is applied at each step).
6. These variables define the nodal masses in the three global directions,  $X$ ,  $Y$ , and  $Z$ . They are to be given in weight units in conjunction with the appropriate acceleration due to gravity, *AGRAV*, that is then used to convert them to masses. These nodal masses are used to compute the dynamic forces on the structure from earthquakes for which the acceleration records are to be provided in input files *for090*, *for091*, and *for092*.
7. This variable defines the identity of the column that is to be associated with the joint at the current node point. The associated column web and flange dimensions along with the depths of beams framing into the joint are used to compute the depth, width and the height of the joint. These dimensions are then used to determine the locations of the attachment points associated with the current node-point at which beams and columns converge. This variable also defines the panel zone coordinate system at the

start of the analysis. The panel zone coordinate system starts out as being identical to the associated column element local coordinate system. As the analysis proceeds, the attachment point coordinates are updated and further updates of the panel zone coordinate system are based on these updated attachment point coordinates.

For gravity columns not connected to any beams, *IDASSCOL* should be assigned a value of 0. For gravity columns that have gravity beams framing into them, specify a non-zero *IDASSCOL*. In this case, clear length of the column will be taken for stiffness computation.

Note that in the output file, *for006*, *DOFX*, *DOFY* & *DOFZ*, refer to the global axes, while *DOFRX*, *DOFRYB*, *DOFRYC*, *DOFRZB*, and *DOFRZC*, refer to the panel zone local axes. If, however, *IDASSCOL* of a particular node is 0, then there cannot be any panel zones associated with that node. Hence, *DOFRYC* and *DOFRZC* are meaningless and the rotations now refer to the global axes, i.e., in this case, the panel zone coordinate system is synonymous with the global coordinate system.

### 4.1.5 Spring Element Data

#### Format

Prepare *NSPR* lines of data, one for each spring, to define the data pertaining to springs in the structural model in the following form:

**NODE DOF SPRSTF**

#### Description

Variable	Field	Notes	Definition
<i>NODE</i>	1	(1)	Identity of the node-point at which the spring is to be located.
<i>DOF</i>	2	(1)	The degree-of-freedom (1-8) the motion of which is resisted by the spring.
<i>SPRSTF</i>	3	(1)	Spring stiffness

#### Notes

1. Springs can be added to the structural model at any node to resist a translational or rotational degree of freedom (DOF) at that node-point. Multiple springs can be located at each node. Springs can be used to model elastic foundations. They can also be used to retrieve reaction forces and moments at support points. In this case, the spring constant/stiffness (*SPRSTF*) should be specified to be much greater (atleast four orders of magnitude greater) than the stiffness of the other elements in the model.



#### 4.1.6 Beam Element Data

##### Format

Prepare  $NPHEL + NFIBEL$  lines of data, one for each beam element, to define the data pertaining to plastic hinge and elastofiber beam elements in the structural model in the following form:

**NODE1 NODE2 IBC IDES EE HSH SIGY FAC1 FAC2 ALPHA IELTYPE  
OLFSRATIO ES SIGU EPSS EPSU ICATFAFRAC ICATFYFRAC**

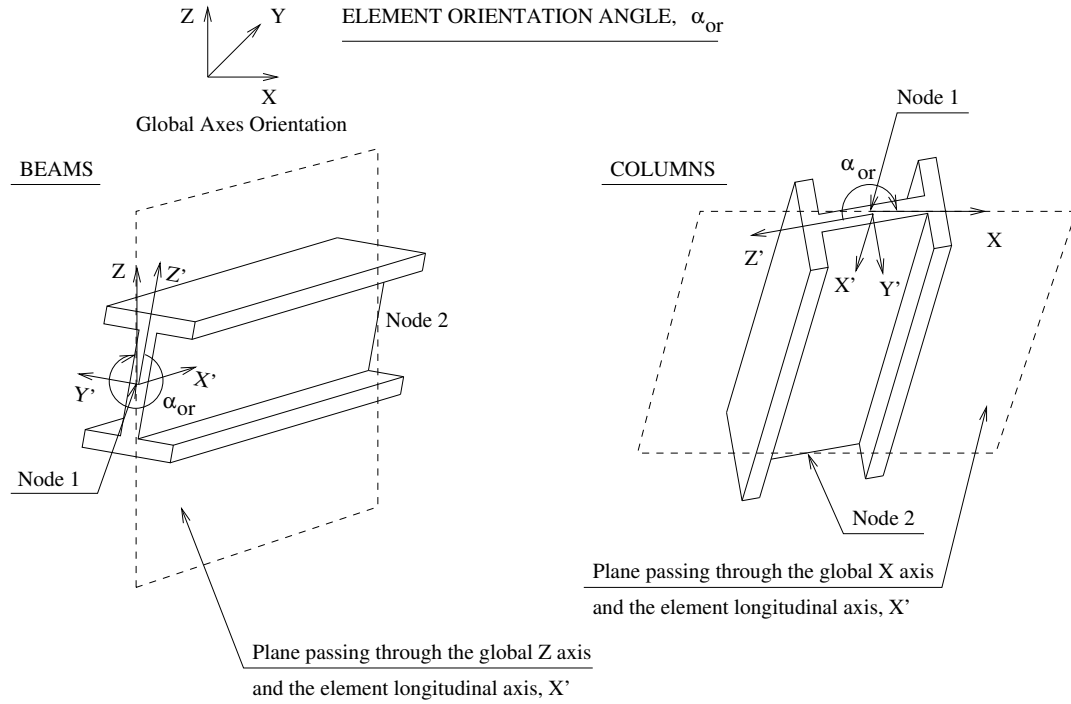
##### Description

Variable	Field	Notes	Definition
<i>NODE1</i>	1	(-)	Node number at end <i>I</i> of the element.
<i>NODE2</i>	2	(-)	Node number at end <i>J</i> of the element.
<i>IBC</i>	3	(1)	Beam or column identifier: 1 for Beam 2 for Column.
<i>IDES</i>	4	(2)	Section designation.
<i>EE</i>	5	(3)	Modulus of Elasticity.
<i>HSH</i>	6	(4)	Strain-hardening modulus.
<i>SIGY</i>	7	(5)	Yield stress.
<i>FAC1</i>	8	(6)	Degree of continuity at end <i>I</i> .
<i>FAC2</i>	9	(6)	Degree of continuity at end <i>J</i> .
<i>ALPHA</i>	10	(7)	Element orientation angle.
<i>IELTYPE</i>	11	(-)	Element type: 0 for plastic-hinge, 1 for elastofiber.
<i>OLFSRATIO</i>	12	(8)	Ratio of fiber segment length to element total length.
<i>ES</i>	13	(9)	Fiber tangent modulus (slope of stress-strain curve) for an elastofiber element at onset of strain-hardening.
<i>SIGU</i>	14	(10)	Fiber ultimate stress for an elastofiber element.
<i>EPSS</i>	15	(11)	Fiber strain at onset of strain-hardening for an elastofiber element.
<i>EPSU</i>	16	(12)	Fiber ultimate strain for an elastofiber element.
<i>ICATFAFRAC</i>	17	(13)	Category for fiber area reduction.
<i>ICATFYFRAC</i>	18	(14)	Category for fiber fracture.

##### Notes

1. A beam element needs to be identified specifically as a beam or a column as the element orientation angle, *ALPHA*, is defined differently for beams and columns.

2. This variable is the label that identifies the section properties listed in input section properties file, *for009*. It should match with one of the numbers listed in the first column of that file.
3. This variable defines the Young's modulus and is used for both plastic hinge and elastofiber elements.
4. This variable defines the post-yield slope of the moment-rotation relationship for the plastic hinge element at the hinge location as a fraction of  $\frac{6EI}{L}$ , i.e., the post-yield slope is taken to be  $HSH * \frac{6EI}{L}$ . Not used for elastofiber element, input 0.0.
5. This variable defines the material yield stress and is used for both plastic hinge and elastofiber elements.
6. To model pinned-end conditions, the plastic moment capacity,  $M_p$  needs to be set to zero. This variable defines the degree of continuity at the end of the element. It should be set to 0.0 if the member end is pinned and 1.0 if the member end is continuous.
7. This variable defines the element orientation angle. It is defined differently for beams and columns as illustrated on Figure 4.1. For columns,  $\alpha$  is the angle made by the  $Z'$  axis with the global  $XZ$  plane, measured using the right-hand-thumb rule with the thumb pointing in the positive direction of the  $X'$  axis. For beams,  $\alpha$  is the angle made by the  $Z'$  axis with a global vertical plane passing through the longitudinal axis of the element,  $X'$ , measured using the right-hand-thumb rule with the thumb pointed in the positive direction of  $X'$  axis. Note that as the analysis proceeds, this angle is updated using the average incremental twist (torsional deformation) in the element at each iteration of each time step.
8. This variable defines the ratio of the length of the end (fiber) segment to the total clear span of the element for an elastofiber element. Based on calibration studies on beams in double-curvature, it is recommended that for materials with low ultimate stress to yield stress ratios (less than 1.4) such as A572-Grade 50 steels, end segment lengths of 3% of the clear span be used while for materials with high ultimate stress to yield stress ratios (greater than 1.4) such as A36 steels, end segment lengths of 11% of the clear span be used. For cantilever beams (in single curvature), these values may be doubled. This variable is not used for a plastic hinge element, input 0.0.



**BEAMS:**  $\alpha_{or}$  = Angle made by the  $Z'$  axis with a plane passing through the global Z axis and the element longitudinal axis,  $X'$ ; it is measured using the right-hand-thumb rule with the thumb pointing in the positive direction of the  $X'$  axis, which runs from node 1 to node 2 of the element

**COLUMNS:**  $\alpha_{or}$  = Angle made by the  $Z'$  axis with a plane passing through the global X axis and the element longitudinal axis,  $X'$ ; it is measured using the right-hand-thumb rule with the thumb pointing in the positive direction of the  $X'$  axis, which runs from node 1 to node 2 of the element

Figure 4.1: Beam/Column Element Orientation Angle

9. This variable defines the slope of the axial stress-strain curve for a fiber in an elastofiber element at the onset of strain hardening (Figure 3.9). A reasonable value for this modulus is 2% of the elastic modulus in the case of structural steel. It is not used for a plastic hinge element, input 0.0.
10. This variable defines the ultimate axial stress for a fiber in an elastofiber element (Figure 3.9). It is not used for a plastic hinge element, input 0.0.
11. This variable defines the strain in the fiber of an elastofiber element at the onset of strain-hardening (Figure 3.9). A reasonable value for this strain is 1.2% in the case of structural steel. It is not used for a plastic hinge element, input 0.0.
12. This variable defines the ultimate strain in the fiber of an elastofiber element (Figure 3.9). A reasonable value for this strain is 16.0% in the case of structural steel. It is not used for a plastic hinge element, input 0.0.
13. To model partial penetration welds or other forms of connections that lead to reduced moment capacities than dictated by gross properties of flanges and webs, it may be necessary to reduce the area of fibers in the flanges and/or webs. This can be done by classifying all the elastofiber elements into fiber area reduction categories, with each category representing a certain reduction pattern in the 20 fibers of each segment of an elastofiber element. This variable defines the fiber area reduction category that the current elastofiber element belongs to. It is not used for a plastic hinge element, input 0.
14. In some moment connections, premature fracture could lead to failure in beam elements prior to the rupture stress being reached. To model this mode of failure, it may be necessary to specify a fracture stress for each fiber based on an assumed probability distribution. This can be done by classifying all the elastofiber elements into different fracture stress distribution categories, with each category representing a certain probability distribution of fracture stresses for the fibers of each segment of an elastofiber element. This variable defines the fracture stress distribution category that the current elastofiber element belongs to. It is not used for a plastic hinge element, input 0.

#### 4.1.7 Categories for Elastofiber Element Fiber Area Reduction

##### Format

Prepare required lines of data, followed by a line of zeros, to define the fiber area reduction categories for elastofiber beam elements in the structural model in the following form:

**ICAT IDSEG IDFIB(20) FAFRAC**

##### Description

Variable	Field	Notes	Definition
<i>ICAT</i>	1	(1)	Fiber area reduction category number.
<i>IDSEG</i>	2	(1,2)	Fiber segment 1 or 2.
<i>IDFIB</i> (20)	3-22	(1,3)	Fiber numbers.
<i>FAFRAC</i>	23	(1,4)	Fraction of fiber area to be used in the analysis (0.0-1.0).

##### Notes

1. This variable identifies the category for which data is provided on this line. Note that each category could have multiple lines of data. For example, the first 6 lines of this data set could belong to category 1, with the first line being applicable to segment 1 top flange fibers, the second line being applicable to segment 1 bottom flange fibers, the third line being applicable to segment 1 web fibers, with the fourth, fifth and sixth lines being applicable to the corresponding data for segment 2. Many such combinations are possible for each category. This is why this data block needs to be followed by a line of zeros.
2. This variable defines the segment number (1 or 2) of the elastofiber element for which this line of data is applicable.
3. Provide 20 numbers corresponding to the fibers of the segment, *IDSEG*, for which the fiber area reduction data provided on this line is applicable. If *FAFRAC* data on this line is applicable to fewer than 20 fibers, then pad the fiber numbers with zeros such that 20 numbers are provided covering Fields 3-22 of the data line.
4. This variable defines the fraction of fiber area to be used for the fibers specified in *IDFIB*(20) of the segment, *IDSEG*, for elastofiber elements belonging to the current

category, *ICAT*.

#### 4.1.8 Seed for Random Number Generation for Elastofiber Element Fiber Fracture Stress Computation

##### Format

Prepare one line of data to define the seed for random number generation to be used for picking a probabilistic fracture stress for fibers of an elastofiber element in the following form:

**ISEED**

##### Description

Variable	Field	Notes	Definition
<i>ISEED</i>	1	(1)	Seed to generate random numbers.

##### Notes

1. This variable is used as a seed to generate a random number for each line of data of the fiber area reduction category of each elastofiber element which is then used to pick one of the 10 fracture stresses listed on that line of that category to be used for all the fibers listed on that line of that category.

#### 4.1.9 Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution

##### Format

Prepare required lines of data, followed by a line of zeros, to define the fiber fracture probability distribution categories for elastofiber beam elements in the structural model in the following form:

**ICAT IDSEG IDFIB(20) FYFRAC(10)**

##### Description

Variable	Field	Notes	Definition
<i>ICAT</i>	1	(1)	Fiber fracture stress probability distribution category number.
<i>IDSEG</i>	2	(1,2)	Fiber segment 1 or 2.
<i>IDFIB</i> (20)	3-22	(1,3)	Fiber numbers.
<i>FYFRAC</i> (10)	23-32	(1,4)	Fracture stress probability distribution.

##### Notes

1. This variable identifies the category for which data is provided on this line. Note that each category could have multiple lines of data. For example, the first 6 lines of this data set could belong to category 1, with the first line being applicable to segment 1 top flange fibers, the second line being applicable to segment 1 bottom flange fibers, the third line being applicable to segment 1 web fibers, with the fourth, fifth and sixth lines being applicable to the corresponding data for segment 2. Many such combinations are possible for each category. This is why this data block needs to be followed by a line of zeros.
2. This variable defines the segment number (1 or 2) of the elastofiber element for which this line of data is applicable.
3. Provide 20 numbers corresponding to the fibers of the segment, *IDSEG*, for which the fiber fracture stress distribution data provided on this line is applicable. If *FYFRAC* data on this line is applicable to fewer than 20 fibers, then pad the fiber numbers with zeros such that 20 numbers are provided covering Fields 3-22 of the data line.



4. Provide 10 numbers specifying the fracture stress distribution for the fibers specified in  $IDFIB(20)$  of the segment,  $IDSEG$ , for elastofiber elements belonging to the current category,  $ICAT$ . For each line of data in this data block, the following steps are followed:

For each elastofiber element with  $ICATFYFRAC$  that is identical to the current category,  $ICAT$ , a random number (1-10) is generated.

The fracture stress listed in  $FYFRAC(10)$  corresponding to this random number is then used as the fracture stress for all the fibers specified in  $IDFIB(20)$  of the segment,  $IDSEG$ , for that element.

#### 4.1.10 Panel Zone Element Data

##### Format

Prepare  $NPZ0 + NPZ1$  lines of data, one for each panel zone, to define the data pertaining to panel zone elements in the structural model in the following form:

**NODE NPZZORY IPZTYPE G HSH TAUY THK DTHK HPZ**

##### Description

Variable	Field	Notes	Definition
<i>NODE</i>	1	(1)	Node number.
<i>NPZZORY</i>	2	(2)	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
<i>IPZTYPE</i>	3	(3)	Panel zone type: 0 for bilinear and 1 for linear-quadratic.
<i>G</i>	4	(4)	Shear modulus.
<i>HSH</i>	5	(5)	Strain hardening modulus.
<i>TAUY</i>	6	(6)	Shear yield stress.
<i>THK</i>	7	(7)	Panel zone thickness.
<i>DTHK</i>	8	(8)	Doubler plate thickness.
<i>HPZ</i>	9	(9)	Panel zone height.

##### Notes

1. Each node can have upto 2 panel zones in two orthogonal directions. This variable defines the node number at which a panel zone is to be located. Note that the nodal restraints specified for this node in the nodal data block must suitably reflect the existence of a panel zone (i.e., with either the *DOFRY* variable or the *DOFRZ* variable being assigned a value of 2).
2. This variable defines the orientation of the panel zone whether it is in the  $X'Z'$  plane of the associated column of this joint (ACJ), corresponding to the major axis bending of the ACJ or whether it is in the  $X'Y'$  plane of the associated column of this joint (ACJ), corresponding to the minor axis bending of the ACJ. Assign a value of 1 for panel zone in the  $X'Z'$  plane of ACJ and a value of 2 for panel zone in the  $X'Y'$  plane of ACJ.
3. This variable defines the shear stress-strain law for the panel zone material. The

stress-strain relation could be bilinear (Figure 3.2) in which case, this variable should be assigned a value of 0, or linear-quadratic (Figure 3.3) in which case, this variable should be assigned a value of 1.

4. This variable defines the elastic shear modulus of the panel zone material. For steel, the shear modulus can be taken to be  $G = \frac{E}{2.5}$ .
5. This variable defines the post-yield slope of the shear stress-strain relationship for a bilinear panel zone as a fraction of  $G$ , i.e., the post-yield slope is taken to be  $HSH * G$ . Not used for elastofiber element, input 0.0.
6. This variable defines the shear stress in the panel zone at which yield occurs. Per the Von-Mises yield criterion, the shear yield stress can be taken to be  $\tau_y = \frac{\sigma_y}{\sqrt{3}}$ , where  $\sigma_y$  is the tensile yield stress of the material.
7. This variable defines the thickness of the panel zone to be used for the analysis. Assign a negative number to this variable if the web/flange thickness of the ACJ is to be used. If NPZZORY is 1, then the sum of the thicknesses of all the webs of the ACJ will be used and if NPZZORY is 2, then the sum of the thicknesses of all the flanges of the ACJ will be used.
8. This variable defines the thickness of any doubler plates that have to be used to reinforce the joint.
9. This variable defines the height of the panel zone to be used in the analysis. Assign a value of the negative of the beam ID if the panel zone height needs to be assigned the depth of a certain beam. Thus, if this variable was assigned an integer value of -500, then, the height of the panel zone would be taken to be equal to the depth of beam element, 500.

#### 4.1.11 Diaphragm Element Control Data

##### Format

Prepare one line of data, giving the control data for elastic diaphragm elements in the structural model in the following form:

**NUMPLSTREL NUMPLSTRMAT NINTPTPLSTR NPLSTRSS**

##### Description

Variable	Field	Notes	Definition
<i>NUMPLSTREL</i>	1	(-)	Number of diaphragm elements.
<i>NUMPLSTRMAT</i>	2	(-)	Number of diaphragm element material types.
<i>NINTPTPLSTR</i>	3	(1)	Number of integration points for Gauss quadrature.
<i>NPLSTRSS</i>	4	(2)	Number of diaphragm elements for which principal stress-strain history is required.

##### Notes

1. This variable defines the number of integration points to be used for Gauss quadrature of a diaphragm element. If a 1-point Gauss quadrature is to be used, assign a value of 1 to this variable. If a 2x2 Gauss quadrature with 4 integration points is to be used, assign a value of 2 to this variable.
2. This variable defines the number of diaphragm elements for which principal stress-strain history is required. Output is presented in file, *PLSTRSS*, and includes the two principal stresses, the angle of the principal plane and the maximum shear stress in the element at each time step of the analysis.

### 4.1.12 Diaphragm Element Material Data

#### Format

Prepare *NUMPLSTRMAT* lines of data, one for each diaphragm element material type, to define the data pertaining to diaphragm element materials in the structural model in the following form:

**E POIS**

#### Description

Variable	Field	Notes	Definition
<i>E</i>	1	(1)	Elastic modulus.
<i>POIS</i>	2	(1)	Poisson's ratio.

#### Notes

1. For each of the *NUMPLSTRMAT* materials, provide the elastic modulus and the Poisson's ratio. For concrete slabs, the elastic modulus can be taken to be  $57000\sqrt{f'_c}$  while the Poisson's ratio can be taken to be 0.30.

#### 4.1.13 Diaphragm Element Connectivity Data

##### Format

Prepare required number of lines of data, to generate the node connectivity of diaphragm elements and assign their material properties in the following form. End with a line of zeros.

**N NE NG MAT THK N1 N2 N3 N4 LMG**

##### Description

Variable	Field	Notes	Definition
<i>N</i>	1	(1,4)	First element in the series to be generated.
<i>NE</i>	2	(1,4)	Last element in the series to be generated.
<i>NG</i>	3	(1,4)	Increment in diaphragm element number.
<i>MAT</i>	4	(2)	Material number.
<i>THK</i>	5	(3)	Diaphragm element thickness.
<i>N1</i>	6	(1,4)	Node 1 of first element in the series to be generated.
<i>N2</i>	7	(1,4)	Node 2 of first element in the series to be generated.
<i>N3</i>	8	(1,4)	Node 3 of first element in the series to be generated.
<i>N4</i>	9	(1,4)	Node 4 of first element in the series to be generated.
<i>LMG</i>	10	(5)	Increment to be applied to the node numbers <i>N1-N4</i> for the next element in the series to be generated.

##### Notes

1. For diaphragm elements,  $N$ ,  $N + NG$ ,  $N + NG * 2, \dots$ ,  $NE - NG$ , and  $NE$ , the element material set is set to  $MAT$  and the thickness is set to  $THK$ .
2. This variable defines the ID of the material property set that needs to be assigned to elements,  $N$ ,  $N + NG$ ,  $N + NG * 2, \dots$ ,  $NE - NG$ , and  $NE$ . The material property sets are defined in the Plane Stress Element Material Data block.
3. This variable defines the thickness of diaphragm elements,  $N$ ,  $N + NG$ ,  $N + NG * 2, \dots$ ,  $NE - NG$ , and  $NE$ .
4. The nodal connectivity for elements,  $N$ ,  $N + NG$ ,  $N + NG * 2, \dots$ ,  $NE - NG$ , and  $NE$ , is taken to be  $[N1, N2, N3, N4]$ ,  $[N1 + LMG, N2 + LMG, N3 + LMG, N4 + LMG]$ ,  $[N1 + LMG * 2, N2 + LMG * 2, N3 + LMG * 2, N4 + LMG * 2], \dots$ ,  $[N1 + LMG * (\frac{NE - N}{NG} - 1), N2 + LMG * (\frac{NE - N}{NG} - 1), N3 + LMG * (\frac{NE - N}{NG} - 1), N4 + LMG * (\frac{NE - N}{NG} - 1)]$ , and  $[N1 + LMG * \frac{NE - N}{NG}, N2 + LMG * \frac{NE - N}{NG}, N3 + LMG * \frac{NE - N}{NG}, N4 + LMG * \frac{NE - N}{NG}]$

respectively. Note that  $N1$ ,  $N2$ ,  $N3$ , and  $N4$  need to be defined in a cyclic fashion, otherwise, a negative Jacobian will result in abrupt termination of the program.

#### 4.1.14 Diaphragm Element Output Control Data

##### Format

Prepare required number of lines of data, to specify the plane stress elements for which principal stress-strain histories are required, in the following form:

**N1 N2**

##### Description

Variable	Field	Notes	Definition
<i>N1</i>	1	(1)	First diaphragm element in the series for which principal stress-strain histories are required.
<i>N2</i>	2	(1)	Last diaphragm element in the series for which principal stress-strain histories are required.

##### Notes

1. These variables define the series of diaphragm elements for which time-histories of principal stresses are required. Provide as many lines as required. The total number of elements listed here must add up to the input variable, *NPLSTRSS*, specified in the Plane Stress Element Control Data block.



#### 4.1.15 Time History Output Control Data

##### Format

Prepare *NRTH* lines of data, specifying the nodal and element response quantities for which time-histories are required, in the following form:

**NODE RESPID ELEM RESPID PZ RESPID ELEM SEG FIB RESPID**

##### Description

Variable	Field	Notes	Definition
<i>NODE</i>	1	(1)	Node number.
<i>RESPID</i>	2	(2)	Nodal response quantity.
<i>ELEM</i>	3	(3)	Beam element number.
<i>RESPID</i>	4	(4)	Element response quantity.
<i>NODE</i>	5	(5)	Node number.
<i>RESPID</i>	6	(6)	Panel zone response quantity.
<i>ELEM</i>	7	(7)	Elastofiber type beam element number.
<i>SEG</i>	8	(8)	Segment number.
<i>FIB</i>	9	(9)	Fiber number.
<i>RESPID</i>	10	(10)	Fiber response quantity.

##### Notes

1. This variable defines the node number for which the time-history of a nodal response quantity, *RESPID*, is required to be output in files *for006* and *RTH*. Assign a value of zero if a nodal response quantity is not required.
2. This variable defines the nodal response quantity of *NODE*, the time-history of which is required to be output in files *for006* and *RTH*. The various nodal response quantities and the corresponding values for *RESPID* are listed below. Assign a value of zero if a nodal response quantity is not required.

Input	Response Quantity
0	If element or panel zone or fiber response quantity is required.
1	$U$ (Global $X$ direction) DOF displacement.
2	$V$ (Global $Y$ direction) DOF displacement.
3	$W$ (Global $Z$ direction) DOF displacement.
4	$\theta_{\bar{X}}$ (Panel zone coordinate system $\bar{X}$ direction) DOF rotation.
5	$\theta_{\bar{Y}}$ (Panel zone coordinate system $\bar{Y}$ direction) DOF rotation.
7	$\theta_{\bar{Z}}$ (Panel zone coordinate system $\bar{Z}$ direction) DOF rotation.
11	Elastic spring internal force.

3. This variable defines the beam element number for which the time-history of an element response quantity, *RESPID*, is required to be output in files *for006* and *RTH*. Assign a value of zero if a beam element response quantity is not required.

4. This variable defines the beam element response quantity of *ELEM*, the time-history of which is required to be output in files *for006* and *RTH*. The various beam element response quantities and the corresponding values for *RESPID* are listed below. Assign a value of zero if a beam element response quantity is not required.

Input	Response Quantity
0	If nodal or panel zone or fiber response quantity is required.
1	Moment, $M_{X'}$ , about element local $X'$ axis (Torsion).
2	Moment, $M_{Y'}^1$ , at node 1 about element local $Y'$ axis (Major axis bending).
3	Moment, $M_{Y'}^2$ , at node 2 about element local $Y'$ axis (Major axis bending).
4	Moment, $M_{Z'}^1$ , at node 1 about element local $Z'$ axis (Minor axis bending).
5	Moment, $M_{Z'}^2$ , at node 2 about element local $Z'$ axis (Minor axis bending).
6	Kink angle, $\kappa_{Y'}^1$ , at node 1 about element local $Y'$ axis (Major axis bending).
7	Kink angle, $\kappa_{Y'}^2$ , at node 2 about element local $Y'$ axis (Major axis bending).
8	Kink angle, $\kappa_{Z'}^1$ , at node 1 about element local $Z'$ axis (Minor axis bending).
9	Kink angle, $\kappa_{Z'}^2$ , at node 2 about element local $Z'$ axis (Minor axis bending).
10	Axial force, $P$ , in element.
11	Axial strain, $\epsilon$ , in element.
12	Updated length, $L$ , of element.

5. This variable defines the node number corresponding to the panel zone element for which the time-history of a panel zone response quantity, *RESPID*, is required to be output in files *for006* and *RTH*. Assign a value of zero if a panel zone element response quantity is not required.

6. This variable defines the response quantity of the panel zone element at *NODE*, the time-history of which is required to be output in files *for006* and *RTH*. The various

panel zone element response quantities and the corresponding values for *RESPID* are listed below. Assign a value of zero if a panel zone element response quantity is not required.

Input	Response Quantity
0	If nodal or beam element or fiber response quantity is required.
1	Moment, $M_{\bar{Y}}$ , about panel zone element local $\bar{Y}$ axis in the ACJ $X'Z'$ panel zone (corresponding to the major axis bending of the ACJ).
2	Shear strain, $\gamma_{\bar{Y}}$ , about panel zone element local $\bar{Y}$ axis in the ACJ $X'Z'$ panel zone (corresponding to the major axis bending of the ACJ).
3	Moment, $M_{\bar{Z}}$ , about panel zone element local $\bar{Z}$ axis in the ACJ $X'Y'$ panel zone (corresponding to the minor axis bending of the ACJ).
4	Shear strain, $\gamma_{\bar{Z}}$ , about panel zone element local $\bar{Z}$ axis in the ACJ $X'Y'$ panel zone (corresponding to the minor axis bending of the ACJ).

7. This variable defines the elastofiber beam element number for which the time-history of a fiber response quantity, *RESPID*, is required to be output in files *for006* and *RTH*. Assign a value of zero if an elastofiber beam element fiber response quantity is not required.
8. This variable defines the fiber segment ID (1 or 2) that contains the fiber for which a fiber response quantity, *RESPID*, is required to be output in files *for006* and *RTH*. Assign a value of zero if an elastofiber beam element fiber response quantity is not required.
9. This variable defines the fiber ID (1-20, Figure 3.7) for which a response quantity, *RESPID*, is required to be output in files *for006* and *RTH*. Assign a value of zero if an elastofiber beam element fiber response quantity is not required.
10. This variable defines the fiber response that is required to be output in files *for006* and *RTH*. Assign a value of zero if an elastofiber beam element fiber response quantity is not required. Assign a value of 1 if fiber stress history is required or a value of 2 if strain history is required.

#### 4.1.16 Ground Acceleration Scaling Factors

##### Format

Prepare one line of data, specifying the ground acceleration scaling factors, in the following form:

**GAMULTX GAMULTY GAMULTZ**

##### Description

Variable	Field	Notes	Definition
<i>GAMULTX</i>	1	(1)	Scaling factor for X-component of ground acceleration.
<i>GAMULTY</i>	2	(1)	Scaling factor for Y-component of ground acceleration.
<i>GAMULTZ</i>	3	(1)	Scaling factor for Z-component of ground acceleration.

##### Notes

1. These variables define the scaling factors to be applied to the ground motion acceleration records provided in input files *for090*, *for091*, and *for092*. They can be used to convert the acceleration records from one system of units to another. Of course, they can be used for direct scaling of records.

#### 4.1.17 Deformed Shape Output Control Data

##### Format

Prepare one line of data, specifying the time interval at which deformed shape nodal coordinates are required, in the following form:

**DTOUT**

##### Description

Variable	Field	Notes	Definition
<i>DTOUT</i>	1	(1)	Time interval at which deformed shape nodal coordinates are required.

##### Notes

1. This variable defines the time interval at which the nodal coordinates of the structure are required as the structure deforms during time-history analysis. Deformed shape nodal coordinate data may require large amounts of storage space and this variable can be used effectively to keep the required storage space in check. Normally, data written at 0.10 seconds should be sufficient to create realistic animations of the structure as it vibrates during, say, an earthquake.

## 4.2 Input File for009

File *for009* is an input data file listing beam element section properties. Beam sections are required to be identified by a positive or a negative integer label. Positive integer labels are reserved for wide-flanged I-sections, whereas, negative integer labels are reserved for box-sections. At the present time, beam cross-sections are limited to wide-flanged and box sections.

### 4.2.1 Labeling

Prepare one line labeling each of the columns in the data file. This line is treated as a comment line and is not read by the program. It has been inserted for user convenience in working with the data in this section property file.

### 4.2.2 Beam Section Data

#### Format

Prepare upto a maximum of 10000 lines of data to define the beam section properties to be used in the structural model in the following form:

**IDES AREA SMI WMI SSM WSM SZ WZ DEPTH WEBTHK FLGWDTH  
FLGTHK TORJ**

**Description**

<b>Variable</b>	<b>Field</b>	<b>Note</b>	<b>Definition</b>
<i>IDES</i>	1	(1)	Section designation (label).
<i>AREA</i>	2	(-)	Area of the cross-section.
<i>SMI</i>	3	(-)	Moment of inertia about the strong (major) axis of the section.
<i>WMI</i>	4	(-)	Moment of inertia about the weak (minor) axis of the section.
<i>SSM</i>	5	(-)	Section modulus about the strong (major) axis of the section.
<i>WSM</i>	6	(-)	Section modulus about the weak (minor) axis of the section.
<i>SZ</i>	7	(-)	Plastic section modulus about the strong (major) axis of the section.
<i>WZ</i>	8	(-)	Plastic section modulus about the weak (minor) axis of the section.
<i>DEPTH</i>	9	(-)	Section depth.
<i>WEBTHK</i>	10	(-)	Web thickness.
<i>FLGWDTH</i>	11	(-)	Flange width.
<i>FLGTHK</i>	12	(-)	Flange thickness.
<i>TORJ</i>	13	(-)	Torsional constant.

**Notes**

1. *IDES* should be an integer. Assign a positive integer for wide-flanged I-sections and a negative integer for box sections.

**4.3 Input Files for090, for091 and for092**

Acceleration histories in the global  $X$ ,  $Y$ , and  $Z$  directions should be provided in input files *for090*, *for091*, and *for092* respectively.

**4.3.1 Heading**

Prepare one line providing the station information, units etc. for the record provided in the data file. This line is treated as a comment line and is not read by the program. It has been inserted for user convenience in identifying the recorded data.

### 4.3.2 Acceleration Data Points

#### Format

Prepare required number of lines of data to define atleast *NDSTPS* data points on the acceleration records. Separate each data point from the other with a blank space. Any number of data points can be provided on each line. The total number of data points should be greater than or equal to the input variable, *NDSTPS*, specified in the Main Control Data block.



## Chapter 5 Program Output

Ten output files are generated from a FRAME3D analysis. These are listed in Table 5.1.

Table 5.1: Output Files from FRAME3D Analysis

<b>File Name</b>	<b>Description</b>
<i>for006</i>	Comprehensive output file.
<i>RTH</i>	Node and element response time-histories.
<i>COORD</i>	Undeformed coordinates of the structure.
<i>DEFCOORD</i>	Deformed coordinates of the structure.
<i>BEAM</i>	Beam element connectivity.
<i>PLSTR</i>	Diaphragm element connectivity.
<i>BMPLROT</i>	Beam element plastic rotations.
<i>PZPLROT</i>	Panel zone element plastic rotations.
<i>PLSTRSS</i>	Diaphragm element principal stress time-histories.
<i>PLSTRSMAX</i>	Diaphragm element peak principal stresses.

The following sections describe the output provided in each file in greater detail.

### 5.1 File for006

*for006* is the main output file that is generated from a FRAME3D analysis. Output includes echoing of the input as well as user required responses in addition to other critical analysis information generated by the program such as global degree of freedom numbers etc. The various sections in the output file are listed in Table 5.2. Notes are provided where new variables are encountered as well as where program generated information is printed out. For all other variables and their definitions, refer to Section 4.1.

Table 5.2: File for006: List of Output Sections

S.No.	Output Section	Notes
1.	Heading	(-)
2.	Main Control Data	(-)
3.	Integration and Iteration Data	(-)
4.	Nodal Data	(-)
5.	Degree of Freedom Numbers	(1)
6.	Spring Element Data	(-)
7.	Joint Dimensions	(2)
8.	Categories for Elastofiber Element Fiber Area Reduction	(-)
9.	Seed for Random Number Generation for Elastofiber Element Fiber Fracture Stress Computation	(-)
10.	Categories for Elastofiber Element Fiber Fracture Stress Probability Distribution	(-)
11.	Beam Element Data	(3)
12.	Panel Zone Element Data	(4)
13.	Diaphragm Element Control Data	(-)
14.	Diaphragm Element Material Data	(-)
15.	Diaphragm Element Connectivity Data	(-)
16.	Diaphragm Element Output Control Data	(-)
17.	Bandwidth Information	(5)
18.	Time History Output Control Data	(-)
19.	Static Solution	(6)
20.	Static Element Responses	(7)
21.	Time History Data	(8)
22.	Dynamic Solution	(9)
23.	Minimum Static Plus Dynamic Element Responses	(10,7)
24.	Maximum Static Plus Dynamic Element Responses	(10,7)
25.	Panel Zone Inelastic Rotations	(11)
26.	Beam End Inelastic Rotations	(12)

### Notes

1. In this section of the output, the total number of degrees of freedom,  $NDOF$ , in the structural model as computed by the program is printed. In addition, the global degrees of freedom numbers from 1 to  $NDOF$  corresponding to the 8 possible degrees of freedom at each node are listed. A zero against a nodal degree of freedom indicates that that degree of freedom is restrained. An identical degree of freedom number for  $DOFRYB$  and  $DOFRYC$  or  $DOFRZB$  and  $DOFRZC$  indicates a rigid panel zone at that node in the corresponding direction, whereas distinct numbers for these indicates the presence of a panel zone with bilinear or linear-quadratic shear stress-

strain behavior.

2. In this section, the height, depth and width of the joint at each node are printed. The height of the joint is computed as the maximum depth of all the beams framing into that joint. The depth and width of the joint are taken to be the same as those of the associated column (ACJ). These dimensions are then used to determine the location of the 6 attachment points associated with the joint at which beams and columns connect. Zero dimensions indicate a free or restrained end, i.e., no beam-column joint exists at this node.
3. In addition to reflecting the input data provided by the user, this section provides some more data that is internally generated by the program. *N1ATTACH* and *N2ATTACH* indicate the attachment point (1-6) at nodes 1 and 2 at which the beam element frames into the two joints. *LENGTH* indicates the clear length of the element between the two attachment points at the two nodes.
4. In addition to reflecting the input data provided by the user, this section provides the total number of panel zones, *NPZ*, computed by the program based on the joint restraint data provided by the user in the Nodal Data block. Note that each node could have a maximum of two panel zones, one in the  $X'Z'$  plane of the ACJ and the other in the  $X'Y'$  plane of the ACJ.
5. This section reflects the allowance provided by the user for the half-bandwidth of the stiffness matrix alongwith the half-bandwidth required for diaphragm elements and beam elements as computed internally by the program. If either of these two number exceeds the user-defined half-bandwidth, a warning is printed and the program is terminated. In such cases, increase the user-defined half-bandwidth to the greater of the program-computed half-bandwidths for plane stress elements and beam elements and rerun the analysis.
6. In this section, information pertaining to the static solution is provided. At each static load-step, the average and maximum number of elastofiber beam element local iterations performed for convergence to the member equilibrium are printed out during each global iteration. After global convergence is achieved for each load-step, the values of the *NRTH* reponse quantities required by the user are printed out. Note

that these time-histories are printed out sequentially in a separate output file, *RTH*, for convenient plotting (See Section 5.2).

7. The responses of all the beam elements and panel zone elements, along with nodal displacements, at the end of the static analysis are provided in this section.

The beam element responses are provided in the following form:

**ELEM MPYL1 MYL1 MPYL2 MYL2 MPZL1 MZL1 MPZL2 MZL2 KAPYL1  
KAPYL2 KAPZL1 KAPZL2 PY P PEPS MXL ALPHA ELEM**

**Description**

<b>Variable</b>	<b>Definition</b>
<i>ELEM</i>	Beam element number.
<i>MPYL1</i>	Plastic moment capacity at node 1 of the element about the local $Y'$ (major) axis.
<i>MYL1</i>	Moment in the element at node 1 about the local $Y'$ (major) axis.
<i>MPYL2</i>	Plastic moment capacity at node 2 of the element about the local $Y'$ (major) axis.
<i>MYL2</i>	Moment in the element at node 2 about the local $Y'$ (major) axis.
<i>MPZL1</i>	Plastic moment capacity at node 1 of the element about the local $Z'$ (minor) axis.
<i>MZL1</i>	Moment in the element at node 1 about the local $Z'$ (minor) axis.
<i>MPZL2</i>	Plastic moment capacity at node 2 of the element about the local $Z'$ (minor) axis.
<i>MZL2</i>	Moment in the element at node 2 about the local $Z'$ (minor) axis.
<i>KAPYL1</i>	Kink rotation in the element at node 1 about the local $Y'$ (major) axis.
<i>KAPYL2</i>	Kink rotation in the element at node 2 about the local $Y'$ (major) axis.
<i>KAPZL1</i>	Kink rotation in the element at node 1 about the local $Z'$ (minor) axis.
<i>KAPZL2</i>	Kink rotation in the element at node 2 about the local $Z'$ (minor) axis.
<i>PY</i>	Axial yield force of the element.
<i>P</i>	Axial force in the element.
<i>PEPS</i>	Axial strain in the element.
<i>MXL</i>	Torsion in the element.
<i>ALPHA</i>	Updated orientation angle of the element.
<i>ELEM</i>	Beam element number.

The panel zone element responses are provided in the following form (Note that data pertaining to the two panel zones at each joint are presented separately) :

**NODE NPZZORY MY M PGAM****Description**

<b>Variable</b>	<b>Definition</b>
<i>NODE</i>	Node number.
<i>NPZZORY</i>	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
<i>MY</i>	Panel zone yield moment.
<i>M</i>	Panel zone moment.
<i>PGAM</i>	Plastic shear strain in the panel zone.

The stresses in beam elements due to the static forces on the structure are provided in the following form:

**ELEM YLMSTRESS ZLMSTRESS PSTRESS TSTRESS**

**Description**

<b>Variable</b>	<b>Definition</b>
<i>ELEM</i>	Beam element number.
<i>YLMSTRESS</i>	Greater of the flexural stresses at the two ends of the element about the local $Y'$ (major) axis.
<i>ZLMSTRESS</i>	Greater of the flexural stresses at the two ends of the element about the local $Z'$ (minor) axis.
<i>PSTRESS</i>	Axial stress in the element.
<i>TSTRESS</i>	Total stress in the element.

The nodal displacements due to the static forces on the structure are provided in the following form:

**NODE XDISP YDISP ZDISP**

**Description**

<b>Variable</b>	<b>Definition</b>
<i>NODE</i>	Node number.
<i>XDISP</i>	Displacement of the node in the global $X$ direction.
<i>YDISP</i>	Displacement of the node in the global $Y$ direction.
<i>ZDISP</i>	Displacement of the node in the global $Z$ direction.

8. In this section, the *NDSTPS* data points of the three components of the ground acceleration specified by the user in input files *for090*, *for091*, and *for092*, are reproduced after being scaled by the user-defined scaling factors *GAMULTX*, *GAMULTY*, and *GAMULTZ*, respectively.
9. In this section, information pertaining to the dynamic solution is provided. At each dynamic time-step, the average and maximum number of elastofiber beam element local iterations performed for convergence to the member equilibrium are printed out during each global iteration. After global convergence is achieved for each time-step, the values of the *NRTH* response quantities required by the user are printed out. Note that these time-histories are printed out sequentially in a separate output file, *RTH*, for convenient plotting (See Section 5.2).
10. The minimum (crest) and maximum (peak) responses of all the beam elements and panel zone elements at the end of the dynamic time-history analysis are provided in

this section (this includes static response). The format and definition of all the output variables are the same as that provided in Note 7.

11. In this section, the plastic rotations (% radians)  $YRPZ$  and  $ZRPZ$ , observed at the two panel zones of each joint during the dynamic time-history analysis, are provided.
12. In this section, the plastic rotations (% radians)  $YROT1$ ,  $ZROT1$ ,  $YROT2$ , and  $ZROT2$ , observed at the two nodes of the beam elements about the major ( $Y'$ ) and minor ( $Z'$ ) axes during the dynamic time-history analysis, are provided.

## 5.2 File RTH

File *RTH* is an output file consisting of *NRTH* columns with each column consisting of *NDSTPS* data points of a user-defined response quantity, specified in the Time History Output Control Data block of input file, *for005*. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the output time-histories of response quantities using a program such as Matlab.

## 5.3 File COORD

File *COORD* is an output file consisting of 22 columns with each column consisting of *NNP* data points corresponding to the *NNP* node points in the structural model. Column 1 gives the node number. Columns 2, 3, and 4 give the nodal coordinates ( $X$ ,  $Y$ , and  $Z$ ). Columns 5 through 22 give the coordinates ( $X$ ,  $Y$  and  $Z$ ) of the 6 attachment points at each node. Thus columns 5, 6, and 7 give the coordinates of attachment point 1 and so on. Note that these coordinates correspond to the undeformed configuration of the structure. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the undeformed shape of the structure.

## 5.4 File DEFCOORD

File *DEFCOORD* is an output file consisting of the coordinates of the node points and the attachment points in the same format as file *COORD*, except that these coordinates are deformed shape coordinates written out at times  $DTOUT$ ,  $2 * DTOUT$ ,  $3 * DTOUT$ , ...,

$(NSSTPS + NDSTPS) * DT$ , where  $DTOUT$  is the time interval at which deformed shape coordinates are to be written out as specified by the user in the Deformed Shape Output Control Data block of input file *for005*.  $DT$  is the time-increment on the acceleration records. Thus, this file consists of  $\frac{(NSSTPS + NDSTPS) * DT}{DTOUT}$  (rounded off to the nearest integer) sets of data with each set consisting of  $NNP$  lines. This file, being fully numerical and consisting only of the data points with no additional characters, can be used to conveniently plot the deformed shape of the structure at each time step. All these plots could then be strung together to create an animation of the structure shaking during an earthquake.

## 5.5 File BEAM

File *BEAM* is an output file consisting of  $NPHEL + NFIBEL$  lines of connectivity data for all the beam elements in the following form:

**ELEM N1 N2 N1ATTACH N2ATTACH**

where  $N1$  and  $N2$  correspond to the two nodes of the element while  $N1ATTACH$  and  $N2ATTACH$  correspond to the attachment points to which the element is physically connected at these nodes. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the undeformed and deformed shapes of the structure.

## 5.6 File PLSTR

File *PLSTR* is an output file consisting of  $NUMPLSTREL$  lines of connectivity data for all the 4-noded diaphragm elements in the following form:

**ELEM N1 N2 N3 N4**

where  $N1$ ,  $N2$ ,  $N3$ , and  $N4$ , correspond to the four nodes of the element in that order. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting the undeformed and deformed shapes of the structure.



## 5.7 File BMPLROT

In this file, the plastic rotations (% radians)  $YROT1$ ,  $ZROT1$ ,  $YROT2$ , and  $ZROT2$ , observed at the two nodes of the beam elements about the major ( $Y'$ ) and minor ( $Z'$ ) axes during the dynamic time-history analysis, are provided in the following form:

**ELEM X-N1ATTACH Y-N1ATTACH Z-N1ATTACH YROT1 ZROT1 X-N2ATTACH Y-N2ATTACH Z-N2ATTACH YROT2 ZROT2**

where  $X - N1ATTACH$ ,  $Y - N1ATTACH$ , and  $Z - N1ATTACH$ , are the final coordinates of the element attachment point at node 1 at the end of the analysis, while,  $X - N2ATTACH$ ,  $Y - N2ATTACH$ , and  $Z - N2ATTACH$ , are the final coordinates of the element attachment point at node 2. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting plastic rotation maps of frames in the structure.

The file consists of  $NPHEL + NFIBEL$  lines of data corresponding to all the beam elements in the structural model.

## 5.8 File PZPLROT

In this file, the plastic rotations (% radians)  $YRPZ$  and  $ZRPZ$ , observed at the two panel zones of each joint during the dynamic time-history analysis, are provided in the following form:

**NODE X Y Z YRPZ ZRPZ**

where  $X$ ,  $Y$ , and  $Z$ , are the coordinates of the node point at the end of the analysis. This file, being fully numerical and consisting only of the data points with no additional characters, can be used for conveniently plotting plastic rotation maps of frames in the structure.

The file consists of  $NNP$  lines of data corresponding to the  $NNP$  node points in the structural model.

## 5.9 File PLSTRSS

This file consists of principal stress-strain histories of the  $NPLSTRSS$  diaphragm elements defined in the diaphragm element output control data block of input file, *for005*. For each

time-step, output is presented in the following form:

**PLSTRELEM**  $\sigma_1$   $\sigma_2$   $\theta_1$   $\tau_{max}$

where *PLSTRELEM* corresponds to one of the diaphragm elements specified in the input file,  $\sigma_1$  and  $\sigma_2$  are the two principal stresses,  $\theta_1$  is the angle of the principal plane (corresponding to  $\sigma_1$ ) and  $\tau_{max}$  is the maximum shear stress in the element at that time-step.

## 5.10 File PLSTRSMAX

This file consists of the maximum and minimum principal stresses in each of the *NUMPLSTREL* diaphragm elements during the course of the time-history analysis. The output is presented in the following form:

**PLSTRELEM**  $\sigma_1^{max}$   $\theta_1^{max}$   $\sigma_2^{min}$   $\theta_2^{min}$   $\tau_{max}$

where *PLSTRELEM* corresponds to one of the *NUMPLSTREL* plane stress elements in the model,  $\sigma_1^{max}$  is the maximum of  $\sigma_1$  in the element for all time and  $\theta_1^{max}$  is the corresponding orientation of the principal plane,  $\sigma_2^{min}$  is the minimum of  $\sigma_2$  in the element for all time and  $\theta_2^{min}$  is the corresponding orientation of the principal plane and  $\tau_{max}$  is the maximum shear stress in the element for all time.

## Chapter 6 Examples

### 6.1 Large Deflection of an Elastic Cantilever Beam

The problem of the large deflection of an elastic cantilever beam with a transversely acting point load at the free end has been analyzed by [13]. It has probably been the “test problem” most commonly used in the examination of finite element procedures for geometrically nonlinear beam and frame analysis. The cantilever beam is illustrated in Figure 6.1. Also illustrated on that figure is the comparison of the numerical solution by [13] using elliptic integrals and the finite element solution using the elastofiber element. The beam is discretized into 10 elastofiber elements. The load,  $P$ , is increased gradually from 0 to 155.74 kN (35 Kips) and the horizontal ( $u$ ) and vertical ( $w$ ) deflections of the free end were computed. These displacements are normalized by the length,  $L$ , and plotted against the load,  $P$ , normalized by  $\frac{EI}{L^2}$ . The close match between the analytical and the FRAME3D solutions illustrates the ability of the elastofiber element to solve large deflection problems accurately.

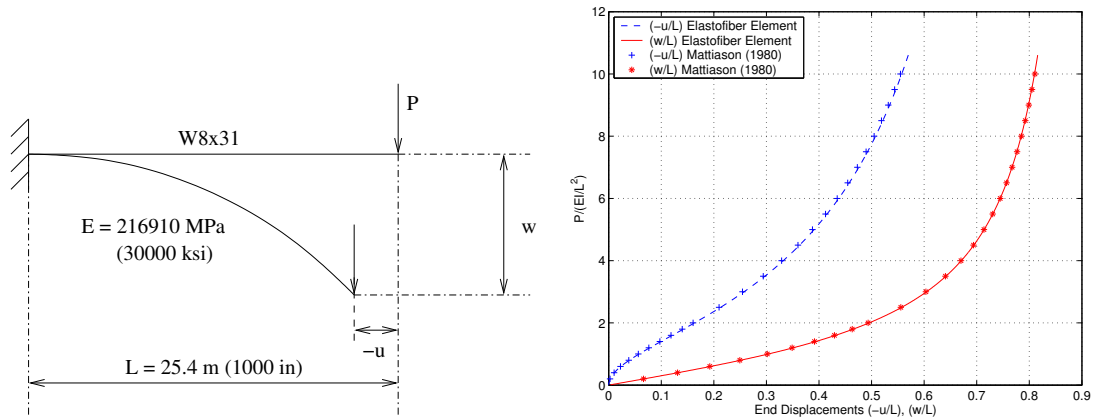


Figure 6.1: Large Deflection of a Cantilever Beam

The input file, *for005*, is shown in Figure 6.2.

```

CHEADING
Cantilever Beam
C NNP NPHEL NFIBEL NPZ0 NPZ1 NSPR MTP NDIM MBD NSSTPS NDSTPS NRTH NGITMAX NLITMAX NOUT
  11 0 10 0 0 0 50 20000 6 1000 0 2 200 50 1
C DT TIMEOSTF BETA GAMMA A0 A1 AGRV STFFACPE STFFACFE STFFACPZ0 STFFACPZ1 TOLF TOLM TOLFIBELF TOLFIBELM THRESH NTHR ITHR
  0.02 0.00 0.25 0.5 0.12566 0.0020372 386.4 0.01 0.00 0.01 0.01 0.01 0.1 0.01 0.1 200. 2 3
C X Y Z XRES YRES ZRES RXRES RYRES RZRES FX FY FZ MX MY MZ IDASSCOL NODE #
  0. 0. 0. 0 0 0 0 0 0 0. 0. 0. 0. 0. 0. 0 1
  100. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 2
  200. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 3
  300. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 4
  400. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 5
  500. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 6
  600. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 7
  700. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 8
  800. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 9
  900. 0. 0. 1 0 1 0 1 0 0. 0.000 0.000 0. 0.000 0 10
  1000. 0. 0. 1 0 1 0 1 0 0. -35.000 0.000 0. 0.000 0 11
C NODE# DOF# SPRING CONSTANT (STIFFNESS) *****ELASTIC SPRING DATA*****
C NODE1 NODE2 IBC IDES EE HSH SIGY FAC1 FAC2 ALPHA IELTYPE OLFSRATIO ES SIGU EPSS EPSU ICATFAFRAC ICATFYFRAC ELEM #
  1 2 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  2 3 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  3 4 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  4 5 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  5 6 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  6 7 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  7 8 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  8 9 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  9 10 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
  10 11 1 8031 30000. 0.100 500000. 1. 1. 0. 1 0.20 30000.00 501000.000 0.80 1.00 1 1 1
C ICAT IDSEG IDFIB(20) FAFRAC (END WITH LINE OF ZEROS)
  1 1 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 1.0
  1 2 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 1.0
  0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
C ISEED (AN INTEGER BETWEEN 1 AND 1000000 TO GENERATE RANDOM NUMBERS FOR FRACTURE STRESS COMPUTATION)
999287
C ICAT IDSEG IDFIB(20) FYFRAC(10) (END WITH LINE OF ZEROS)
  1 1 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  1 2 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
C Note 1: If a -ve THK is input, web/flg thickness of ass. col. of corresponding node will be used for the PZ thickness
C Note 2: To select a particular beam as the beam governing the PZ depth, input the ID of the beam as a negative integer as HPZ
C NODE NPZZORY IPZTYPE G HSH TAU Y THK DTHK HPZ
C NUMPLSTREL NUMPLSTRMAT NINTPTPLSTR(=1 ==> 1-point Quadrature; =2 ==> 2X2 Quadrature) NPLSTRSS
  0 0 20
C E POIS PLSTRMATID (FOR EACH OF THE NUMPLSTRMAT MATERIALS)
C FOR ELEMENTS N+NG, N+NG*2,...., NE-NG, NE, THE ELM MAT SET & THK NUMBERS ARE ALSO SET TO MAT(N) & THK(N) AND
C THE CONNECTIVITY ARRAYS ARE INCREMENTED BY LMG
C 0 0 0 0 0.00 0 0 0 0 0 TO END SERIES
C N NE NG MAT# THK N1 N2 N3 N4 LMG
  0 0 0 0 0.00 0 0 0 0 0
C PRINCIPAL STRESS HISTORIES WILL BE WRITTEN OUT FOR PLANE STRESS ELEMENTS, N1 TO N2
C N1 N2
C NODE RESPONSE: 1-U 2-V 3-W 4-THETAX 5-THETAY 7-THETAZ
C ELEM RESPONSE: 1-Mx 2-My1' 3-My2' 4-Mz1' 5-Mz2' 6-Ky1' 7-Ky2' 8-Kz1' 9-Kz2' 10-P 11-Epsilon
C NODE# RESPID# ELEM# RESPID# PZ# RESPID# ELEM# SEG# FIB# RESPID#
  11 1 0 0 0 0 0 0 0 0
  11 3 0 0 0 0 0 0 0 0
C GAMULTX GAMULTY GAMULTZ
  0.3937 0.0000 0.3937
C TIME INTERVAL AT WHICH DEFORMED SHAPE COORDINATES HAVE TO BE PRINTED OUT (MAX # OF STATES = 250)
0.10

```

Figure 6.2: Large Deflection of a Cantilever Beam: Input file, *for005*

The input file, *for009*, is shown in Figure 6.3.

IDES	AREA	SMI	WMI	SSM	WSM	SZ	WZ	DEPTH	WEBTHK	FLGWIDTH	FLGTHK	TORJ
8031	9.13	110.	37.1	27.5	9.27	30.4	14.1	8.00	0.285	7.995	0.435	0.54

Figure 6.3: Large Deflection of a Cantilever Beam: Input file, *for009*

In this case, since dynamic time-history analysis is not required, input files *for090*, *for091*, and *for092* are not required.

The output file, *for006*, is shown in Figures 6.4 through 6.7.

DATE 05/25/2003 TIME 19:01:30

Cantilever Beam

NNP = 11  
NEL = 10  
NPHEL = 0  
NFIBEL = 10  
NPZO = 0  
NPZI = 0  
NSPR = 0  
MTP = 50  
NDIM = 20000  
MBD = 6  
NSSTPS = 1000  
NDSTPS = 0  
NRTH = 2  
NGITMAX = 200  
NLITMAX = 50  
NOUT = 1  
  
DT = 0.20000E-01  
TIMEOSTF = 0.00000E+00  
BETA = 0.25000E+00  
GAMMA = 0.50000E+00  
A0 = 0.12566E+00  
A1 = 0.20372E-02  
AGRAV = 0.38640E+03  
STFFACPE = 0.10000E-01  
STFFACFE = 0.00000E+00  
STFFACPZO = 0.10000E-01  
STFFACPZI = 0.10000E-01  
TOLF = 0.10000E-01  
TOLM = 0.10000E+00  
TOLFIBELF = 0.10000E-01  
TOLFIBELM = 0.10000E+00  
THRESH = 0.20000E+03  
NTHR = 2  
ITHR = 3

NODAL POINT INPUT

NODE	COORDX	COORDY	COORDZ	IDX	IDY	IDZ	IDTHX	IDTHY	IDTHZ	FX	FY	FZ	MX	MY	MZ	IDASSCOL
1	0.00000E+00	0.00000E+00	0.00000E+00	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
2	0.10000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
3	0.20000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
4	0.30000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
5	0.40000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
6	0.50000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
7	0.60000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
8	0.70000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
9	0.80000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
10	0.90000E+03	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
11	0.10000E+04	0.00000E+00	0.00000E+00	1	0	1	0	0.00000E+00	0.00000E+00	-0.35000E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0

\*\*\*\*\*

NDOF = 30

\*\*\*\*\*

DEGREE OF FREEDOM NUMBERS

NODE	DOFX	DOFY	DOFZ	DOFRX	DOFRYB	DOFRYC	DOFRZB	DOFRZC
1	0	0	0	0	0	0	0	0
2	1	0	2	0	3	3	0	0
3	4	0	5	0	6	6	0	0
4	7	0	8	0	9	9	0	0
5	10	0	11	0	12	12	0	0
6	13	0	14	0	15	15	0	0
7	16	0	17	0	18	18	0	0
8	19	0	20	0	21	21	0	0
9	22	0	23	0	24	24	0	0
10	25	0	26	0	27	27	0	0
11	28	0	29	0	30	30	0	0

NOTE THAT DOFX,Y,& Z REFER TO GLOBAL AXES  
WHILE DOFRX,RBY,RYC,RZB,RZC ALL REFER TO PANEL ZONE  
LOCAL AXES.

IF, HOWEVER, IDASSCOL OF A PARTICULAR NODE IS ZERO,  
THEN THERE CANNOT BE ANY PANEL ZONES ASSOCIATED  
WITH THAT NODE. HENCE, DOFRYC & DOFRZC ARE MEANINGLESS  
AND THE ROTATIONS NOW REFER TO THE GLOBAL AXES  
IN THIS CASE, I.E., THE PANEL ZONE COORDINATE SYSTEM  
IS SYNONYMOUS WITH THE GLOBAL COORDINATE SYSTEM.

FOR GRAVITY COLUMNS THAT DO NOT HAVE ANY BEAMS  
FRAMING INTO THEM, SPECIFY IDASSCOL=0.

FOR GRAVITY COLUMNS THAT HAVE GRAVITY BEAMS FRAMING  
INTO THEM, SPECIFY NON-ZERO IDASSCOL. IN THIS CASE  
CLEAR LENGTH OF COLUMN WILL BE TAKEN FOR STIFFNESS  
COMPUTATIONS.

Figure 6.4: Large Deflection of a Cantilever Beam: Output file, *for006*

```

ELASTIC SPRING DATA

SPRING# NODE# DOF#      SPRING STIFFNESS

JOINT DIMENSIONS

NODE   H   D   W
1 0.0000 0.0000 0.0000
2 0.0000 0.0000 0.0000
3 0.0000 0.0000 0.0000
4 0.0000 0.0000 0.0000
5 0.0000 0.0000 0.0000
6 0.0000 0.0000 0.0000
7 0.0000 0.0000 0.0000
8 0.0000 0.0000 0.0000
9 0.0000 0.0000 0.0000
10 0.0000 0.0000 0.0000
11 0.0000 0.0000 0.0000

ELEMENT INPUT

FIBER ELEMENT AREA REDUCTION CATEGORY INFORMATION
ICAT IDSEG  IDFIB                      FAFRAC
1 1 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01
1 2 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01

ISEED = 999287

FIBER ELEMENT FRACTURE STRESS CATEGORY INFORMATION
ICAT IDSEG  IDFIB                      FYFRAC(1) TO (10)
1 1 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10
1 2 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10

ELEM  NODE1 NIATTACH NODE2 N2ATTACH IBC  IDES  ALPHA  EE  HSH  SIGY  FAC1  FAC2  LENGTH IELTYPE OLFSRAT  ES  SIGU  EPSS  EPSU  ICATFAFRAC ICATFYFRAC
1 1 0 2 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
2 2 0 3 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
3 3 0 4 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
4 4 0 5 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
5 5 0 6 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
6 6 0 7 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
7 7 0 8 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
8 8 0 9 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
9 9 0 10 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1
10 10 0 11 0 1 8031 0.030000E+05 0.1000 0.50000E+06 1.0000 1.0000100.0000 1 0.20 0.30000E+05 0.50100E+06 0.800 1.000 1 1

*****

NPZ = 0

*****

PANEL ZONE INPUT

PZ# NODE Z OR Y IPZTYPE  GG  HSH  TAUY  THK  DTHK  HPZ

PLANE STRESS ELEMENT DATA

NO.OF PLANE STRESS ELEMENTS, NUMPLSTREL = 0
NO.OF PLANE STRESS ELEMENT MATERIALS, NUMPLSTRMAT = 0
NO.OF INTEGRATION POINTS FOR GAUSS QUADRATURE,
NINTPTPLSTR = 2
NO.OF PLANE STRESS ELEMENTS, NUMPLSTRSS, FOR WHICH
PRINCIPAL STRESS HISTORY NEEDS TO BE WRITTEN OUT
IN FILE PLSTRSS = 0

MATERIAL SETS
SET      E      POIS

PLANE STRESS ELEMENT THICKNESSES & MATERIAL SET      NUMBERS AND CONNECTIVITY MATRIX
ELEM  MATSET  THK  NODE1  NODE2  NODE3  NODE4

PLANE STRESS ELEMENTS FOR WHICH PRINCIPAL STRESS
OUTPUT IS REQUIRED (FILE PLSTRSS)

*****

MAX. BANDWIDTH INPUT BY USER      = 6
MAX. BANDWIDTH FOR PLANE STRESS ELEMENTS = 0
MAX. BANDWIDTH FOR FRAME ELEMENTS   = 6
USE THE GREATER OF THE ABOVE TWO BANDWIDTHS

*****

RESPONSE  IDRTH  (1) (2) (3) (4) (5) (6) (7) (8) (9) (10)
R 1      11 1 0 0 0 0 0 0 0 0 0
R 2      11 3 0 0 0 0 0 0 0 0 0

```

Figure 6.5: Large Deflection of a Cantilever Beam: Output file, *for006* (Contd.)

```

STATIC SOLUTION

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 4

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 05/25/2003 TIME 19:01:31

TIME = 0.020 NITER = 4
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
-0.15561E-01 -0.35636E+01

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 4

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 05/25/2003 TIME 19:01:31

TIME = 0.040 NITER = 4
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
-0.46362E-01 -0.71272E+01

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 4

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 05/25/2003 TIME 19:01:32

TIME = 0.060 NITER = 4
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
-0.92387E-01 -0.10690E+02
.
.
.
.

DATE 05/25/2003 TIME 19:07:22

TIME = 19.980 NITER = 11
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
-0.56959E+03 -0.81581E+03

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

DATE 05/25/2003 TIME 19:07:22

TIME = 20.000 NITER = 17
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
-0.56981E+03 -0.81590E+03

```

Figure 6.6: Large Deflection of a Cantilever Beam: Output file, *for006* (Contd.)



```

STATIC ELEMENT RESPONSES

DATE 05/25/2003  TIME 19:07:22

ELEM  MPYL1  MYL1  MPYL2  MYL2  MPZL1  MZL1  MPZL2  MZL2  KAPYL1  KAPYL2  KAPZL1  KAPZL2  PY  P  PEPS  TORSION,MXL  ALPHA  ELEM
1  0.15200E+08 -0.15074E+05 0.15200E+08 0.11672E+05 0.70500E+07 -0.13655E-11 0.70500E+07 -0.44334E-11 -0.90989E-01 0.74140E-01 -0.12213E-16 0.44191E-16 0.45650E+07 0.73849E+01 0.00000E+00 0.22874E-15 0.00000E+00 1
2  0.15200E+08 -0.11672E+05 0.15200E+08 0.87368E+04 0.70500E+07 -0.23332E-11 0.70500E+07 0.34638E-11 -0.70171E-01 0.55655E-01 0.72510E-16 -0.77835E-16 0.45650E+07 0.18874E+02 0.00000E+00 0.65416E-16 0.00000E+00 2
3  0.15200E+08 -0.87369E+04 0.15200E+08 0.64044E+04 0.70500E+07 0.28533E-11 0.70500E+07 -0.61535E-12 -0.52399E-01 0.40871E-01 -0.53554E-16 0.37470E-16 0.45650E+07 0.26016E+02 0.00000E+00 -0.65655E-16 0.00000E+00 3
4  0.15200E+08 -0.64044E+04 0.15200E+08 0.46250E+04 0.70500E+07 -0.10462E-11 0.70500E+07 0.29898E-11 -0.38348E-01 0.29558E-01 0.47811E-16 -0.58579E-16 0.45650E+07 0.30103E+02 0.00000E+00 -0.23644E-16 0.00000E+00 4
5  0.15200E+08 -0.46249E+04 0.15200E+08 0.32914E+04 0.70500E+07 0.15743E-11 0.70500E+07 0.11200E-11 -0.27657E-01 0.21070E-01 -0.14147E-16 -0.10519E-17 0.45650E+07 0.32343E+02 0.00000E+00 -0.44234E-16 0.00000E+00 5
6  0.15200E+08 -0.32915E+04 0.15200E+08 0.22930E+04 0.70500E+07 -0.41678E-12 0.70500E+07 -0.33141E-12 -0.19649E-01 0.14719E-01 0.17984E-17 0.46137E-19 0.45650E+07 0.33538E+02 0.00000E+00 0.58381E-17 0.00000E+00 6
7  0.15200E+08 -0.22931E+04 0.15200E+08 0.15349E+04 0.70500E+07 -0.22804E-12 0.70500E+07 -0.31311E-12 -0.13650E-01 0.99064E-02 0.17658E-17 0.13533E-17 0.45650E+07 0.34166E+02 0.00000E+00 0.39546E-17 0.00000E+00 7
8  0.15200E+08 -0.15350E+04 0.15200E+08 0.94006E+03 0.70500E+07 0.99758E-12 0.70500E+07 0.30864E-12 -0.90837E-02 0.61464E-02 -0.10777E-16 0.44708E-17 0.45650E+07 0.34491E+02 0.00000E+00 -0.78462E-17 0.00000E+00 8
9  0.15200E+08 -0.94012E+03 0.15200E+08 0.44601E+03 0.70500E+07 -0.69139E-13 0.70500E+07 -0.36406E-12 -0.54845E-02 0.30450E-02 -0.29294E-17 0.44199E-17 0.45650E+07 0.34651E+02 0.00000E+00 0.15333E-17 0.00000E+00 9
10 0.15200E+08 -0.44608E+03 0.15200E+08 -0.36930E-01 0.70500E+07 0.69926E-13 0.70500E+07 -0.87044E-14 -0.24736E-02 0.27115E-03 -0.20258E-17 0.13892E-17 0.45650E+07 0.34717E+02 0.00000E+00 0.14589E-18 0.00000E+00 10

NODE NPZZORY  MY  M  PGAM

ELEM YLMSTRESS ZLMSTRESS PSTRESS TSTRESS
1  0.54816E+03 0.47826E-12 0.80886E+00 0.54897E+03
2  0.42443E+03 0.37366E-12 0.20672E+01 0.42650E+03
3  0.31771E+03 0.30780E-12 0.28495E+01 0.32056E+03
4  0.23289E+03 0.52252E-12 0.32972E+01 0.23618E+03
5  0.16818E+03 0.16982E-12 0.35425E+01 0.17172E+03
6  0.11969E+03 0.44960E-13 0.36734E+01 0.12336E+03
7  0.83384E+02 0.33777E-13 0.37422E+01 0.87127E+02
8  0.55817E+02 0.10761E-12 0.37778E+01 0.59594E+02
9  0.34186E+02 0.39272E-13 0.37953E+01 0.37982E+02
10 0.16221E+02 0.75432E-14 0.38026E+01 0.20024E+02

NODE XDISP YDISP ZDISP
1  0.00000E+00 0.00000E+00 0.00000E+00
2 -0.27880E-01 0.00000E+00 -0.21068E+02
3 -0.18942E+02 0.00000E+00 -0.74943E+02
4 -0.52336E+02 0.00000E+00 -0.14920E+03
5 -0.10157E+03 0.00000E+00 -0.23509E+03
6 -0.16354E+03 0.00000E+00 -0.32732E+03
7 -0.23509E+03 0.00000E+00 -0.42291E+03
8 -0.31350E+03 0.00000E+00 -0.52023E+03
9 -0.39656E+03 0.00000E+00 -0.61845E+03
10 -0.48250E+03 0.00000E+00 -0.71709E+03
11 -0.56981E+03 0.00000E+00 -0.81590E+03

```

Figure 6.7: Large Deflection of a Cantilever Beam: Output file, *for006* (Contd.)

## 6.2 2 Story Building with 2 Bays in Each Principal Direction

The second example consists of a 2-story building with 2 bays in either of the two principal directions. The model consists of 27 nodes, 30 beam elements, 8 diaphragm elements and 14 panel zone elements. Out of the 30 beam elements, 4 are gravity columns and are modeled using the plastic-hinge element while the remaining 26 are modeled using the elastofiber elements. The isometric view of the building model with node and beam element numbering is shown in Figure 6.8. The floor plan of the building with column orientations, location of moment framed connections and the diaphragm element numbering is illustrated in Figure 6.9. In addition to gravity loads, the building is subjected to ground shaking from the first 5 seconds of the Tabas strong-motion record from the 1978 Iran earthquake. The records are in units of  $g$  and hence a value of  $386.4 \frac{in}{sec^2}$  is used for the scaling factors, *GAMULTX*, *GAMULTY*, and *GAMULTZ*. Also, note that the variable *TIMEOSTF* has been assigned a value of 4.00 *sec*. Thus, for the first 4 seconds of the record, the program uses the initial elastic stiffness matrix to iterate and after that it uses a tangent stiffness matrix for iteration.

Legend:

(X) Beam-Column Element Number

Y Node Number

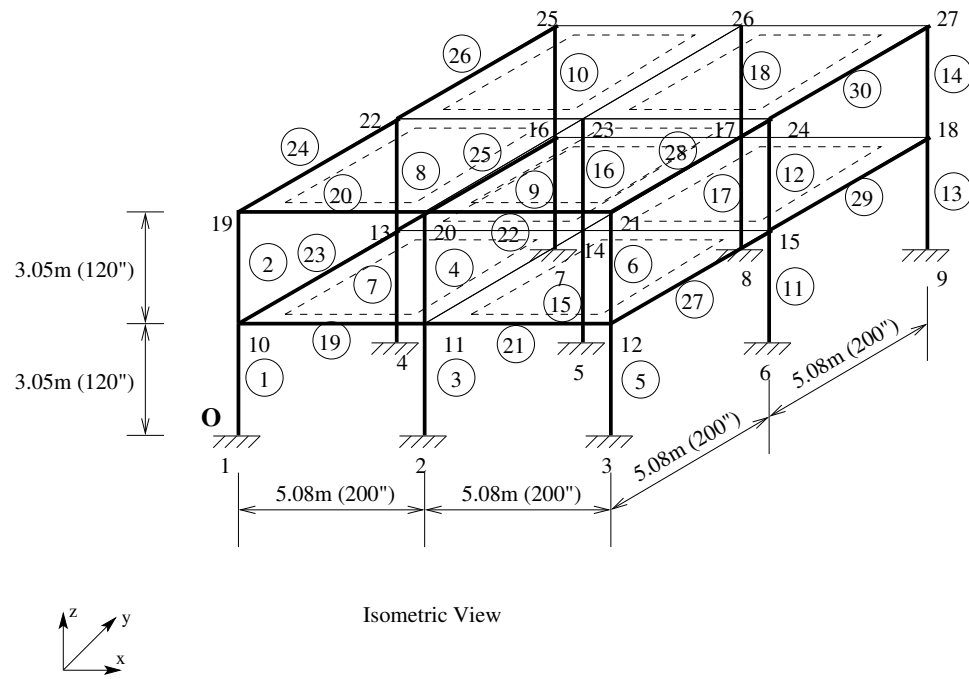
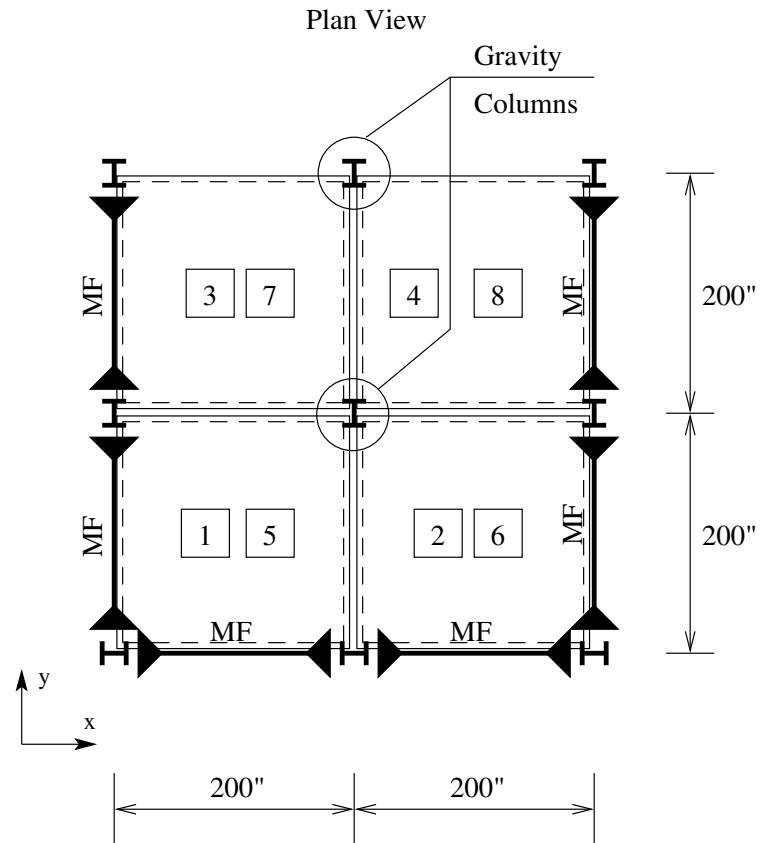


Figure 6.8: Isometric View of 2 Story Building



Legend:

X
---

Y
---

 Plane-stress Element Numbers at 2nd Floor and Roof

Figure 6.9: Plan View of 2 Story Building

The input file, *for005*, is shown in Figures 6.10 through 6.11.

```
CHEADING
2-Storeyed 3D 2-Bay x 2-Bay Building
C NNP NPHL NFBEL NPZ0 NPZ1 NSPR MTP NDIM MBD NSSTPS NDSTPS NRTH NGITMAX NLITMAX NOUT
  27  4  26  0  18  0  40 10000 72  1  1000  3  100  50  1
C DT TIMEOSTF BETA GAMMA A0 A1 AGRV STFFACPHIE STFFACFE STFFACPZ0 STFFACPZ1 TOLF TOLM TOLFIBELF TOLFIBELM THRESH NTHR ITHR
  0.005 4.00  0.25  0.5 0.12566 0.0020372 386.4  0.01  0.02  0.01  0.01 0.00001 0.00001 0.00001 0.00001 200.  2  3
C X  Y  Z  XRES YRES ZRES RXRES RYRES RZRES FX  FY  FZ  MX  MY  MZ  IDASSCOL NODE #
  0.  0.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  1
200.  0.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  2
400.  0.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  3
  0. 200.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  4
200. 200.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  5
400. 200.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  6
  0. 400.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  7
200. 400.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  8
400. 400.  0.  0  0  0  0  0  0  0.  0.  0.  0.  0.  0.  0.  0.  9
  0.  0. 120.  1  1  1  1  2  2  0.  0.-100. 100. 100. 100.  1  10
200.  0. 120.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100.  3  11
400.  0. 120.  1  1  1  1  2  2  0.  0.-100. 100. 100. 100.  5  12
  0. 200. 120.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100.  7  13
200. 200. 120.  1  1  1  1  1  1  0.  0.-100. 100. 100. 100.  0  14
400. 200. 120.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100. 11  15
  0. 400. 120.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100.  9  16
200. 400. 120.  1  1  1  1  1  1  0.  0.-100. 100. 100. 100.  0  17
400. 400. 120.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100. 13  18
  0.  0. 240.  1  1  1  1  2  2  0.  0.-100. 100. 100. 100.  2  19
200.  0. 240.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100.  4  20
400.  0. 240.  1  1  1  1  2  2  0.  0.-100. 100. 100. 100.  6  21
  0. 200. 240.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100.  8  22
200. 200. 240.  1  1  1  1  1  1  0.  0.-100. 100. 100. 100.  0  23
400. 200. 240.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100. 12  24
  0. 400. 240.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100. 10  25
200. 400. 240.  1  1  1  1  1  1  0.  0.-100. 100. 100. 100.  0  26
400. 400. 240.  1  1  1  1  2  1  0.  0.-100. 100. 100. 100. 14  27
C NODE# DOF# SPRING CONSTANT (STIFFNESS) *****ELASTIC SPRING DATA*****
C NODE1 NODE2 IBC IDES  EE  HSH  SIGY  FAC1 FAC2 ALPHA IELTYPE OLFSRATIO ES  SIGU  EPSS  EPSU  ICATFAFRAC ICATFYFRAC ELEM #
  1 10  2 14159 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1  1
10 19  2 14159 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1  2
  2 11  2 14159 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1  3
11 20  2 14159 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1  4
  3 12  2 14159 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1  5
12 21  2 14159 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1  6
  4 13  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1  7
13 22  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1  8
  7 16  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1  9
16 25  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1 10
  6 15  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1 11
15 24  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1 12
  9 18  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1 13
18 27  2 14159 29000. 0.025 50. 1. 1. 90.  1  0.03  580.00 65.000 0.012 0.160  1  1 14
  5 14  2 14159 29000. 0.025 50. 1. 1. 90.  0  0.03  580.00 65.000 0.012 0.160  1  1 15
14 23  2 14159 29000. 0.025 50. 1. 1. 90.  0  0.03  580.00 65.000 0.012 0.160  1  1 16
  8 17  2 14159 29000. 0.025 50. 1. 1. 90.  0  0.03  580.00 65.000 0.012 0.160  1  1 17
17 26  2 14159 29000. 0.025 50. 1. 1. 90.  0  0.03  580.00 65.000 0.012 0.160  1  1 18
10 11  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 19
19 20  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 20
11 12  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 21
20 21  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 22
10 13  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 23
19 22  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 24
13 16  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 25
22 25  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 26
12 15  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 27
21 24  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 28
15 18  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 29
24 27  1 24131 29000. 0.025 50. 1. 1.  0.  1  0.03  580.00 65.000 0.012 0.160  1  1 30
```

Figure 6.10: 2-Story Building: Input file, *for005*

```

C ICAT IDSEG IDFIB(20)                                FAFRAC (END WITH LINE OF ZEROS)
  1  1  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 1.0
  1  2  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 1.0
  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0.0
C ISEED (AN INTEGER BETWEEN 1 AND 1000000 TO GENERATE RANDOM NUMBERS FOR FRACTURE STRESS COMPUTATION)
999287
C ICAT IDSEG IDFIB(20)                                FYFRAC(10) (END WITH LINE OF ZEROS)
  1  1  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  1  2  1  2  3  4  5  6  7  8  9 10 11 12 13 14 15 16 17 18 19 20 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9 9.D9
  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
C Note 1: If a -ve THK is input, web/flg thickness of ass. col. of corresponding node will be used for the PZ thickness
C Note 2: To select a particular beam as the beam governing the PZ depth, input the ID of the beam as a negative integer as HPZ
C NODE NPZZORY IPZTYPE G   HSH  TAUZY  THK   DTHK  HPZ
  10  1  1  11600.  0.05  28.9  -1.0  0.000  -19
  10  2  1  11600.  0.05  28.9  -1.0  0.000  -23
  11  1  1  11600.  0.05  28.9  -1.0  0.000  -21
  12  1  1  11600.  0.05  28.9  -1.0  0.000  -21
  12  2  1  11600.  0.05  28.9  -1.0  0.000  -27
  13  1  1  11600.  0.05  28.9  -1.0  0.000  -23
  15  1  1  11600.  0.05  28.9  -1.0  0.000  -27
  16  1  1  11600.  0.05  28.9  -1.0  0.000  -25
  18  1  1  11600.  0.05  28.9  -1.0  0.000  -29
  19  1  1  11600.  0.05  28.9  -1.0  0.000  -20
  19  2  1  11600.  0.05  28.9  -1.0  0.000  -24
  20  1  1  11600.  0.05  28.9  -1.0  0.000  -20
  21  1  1  11600.  0.05  28.9  -1.0  0.000  -22
  21  2  1  11600.  0.05  28.9  -1.0  0.000  -28
  22  1  1  11600.  0.05  28.9  -1.0  0.000  -24
  24  1  1  11600.  0.05  28.9  -1.0  0.000  -28
  25  1  1  11600.  0.05  28.9  -1.0  0.000  -26
  27  1  1  11600.  0.05  28.9  -1.0  0.000  -30
C NUMPLSTREL NUMPLSTRMAT NINTPTPLSTR(=1 ==> 1-point Quadrature; =2 ==> 2X2 Quadrature) NPLSTRSS
  8  1  20
C E  POIS PLSTRMATID  (FOR EACH OF THE NUMPLSTRMAT MATERIALS)
3600.0 0.3 1
C FOR ELEMENTS N+NG, N+NG*2,..., NE-NG, NE, THE ELM MAT SET & THK NUMBERS ARE ALSO SET TO MAT(N) & THK(N) AND
C THE CONNECTIVITY ARRAYS ARE INCREMENTED BY LMG
C 0 0 0 0 0.00 0 0 0 0 0 TO END SERIES
C N NE NG MAT# THK N1 N2 N3 N4 LMG
  1  2  1  1  3.5 10 11 14 13 1
  3  4  1  1  3.5 13 14 17 16 1
  5  6  1  1  3.5 19 20 23 22 1
  7  8  1  1  3.5 22 23 26 25 1
  0  0  0  0  0.0 0 0 0 0 0
C PRINCIPAL STRESS HISTORIES WILL BE WRITTEN OUT FOR PLANE STRESS ELEMENTS, N1 TO N2
C N1 N2
C NODE RESPONSE: 1-U 2-V 3-W 4-THETAX 5-THETAY 7-THETAZ
C ELEM RESPONSE: 1-Mx 2-My1' 3-My2' 4-Mz1' 5-Mz2' 6-Ky1' 7-Ky2' 8-Kz1' 9-Kz2' 10-P 11-Epsilon
C NODE# RESPID# ELEM# RESPID# PZ# RESPID# ELEM# SEG# FIB# RESPID#
  27  1  0  0  0  0  0  0  0  0
  27  2  0  0  0  0  0  0  0  0
  27  3  0  0  0  0  0  0  0  0
C  GAMULTX  GAMULTY  GAMULTZ
  386.400  386.400  386.400
C TIME INTERVAL AT WHICH DEFORMED SHAPE COORDINATES HAVE TO BE PRINTED OUT (MAX # OF STATES = 250)
0.10

```

Figure 6.11: 2-Story Building: Input file, *for005* (Contd.)

The input file, *for009*, is shown in Figure 6.12.

IDES	AREA	SMI	WMI	SSM	WSM	SZ	WZ	DEPTH	WEBTHK	FLGWDTH	FLGTHK	TORJ
14159	46.43	1900.	748.	254.	96.2	287.	146.0	14.98	0.745	15.565	1.190	19.80
24131	38.5	4020.	340.	329.	53.0	370.	81.5	24.48	0.605	12.855	0.960	9.50
24146	43.0	4580.	391.	371.	60.5	418.	93.2	24.74	0.650	12.900	1.090	13.40
24162	47.7	5170.	443.	414.	68.4	468.	105.0	25.00	0.705	12.955	1.220	18.50

Figure 6.12: 2-Story Building: Input file, *for009*

The output file, *for006*, is shown in Figures 6.13 through 6.23.

DATE 06/20/2003 TIME 19:44:04

## 2-Storeyed 3D 2-Bay x 2-Bay Building

NNP = 27  
 NEL = 30  
 NPHEL = 4  
 NFIBEL = 26  
 NPZ0 = 0  
 NPZ1 = 18  
 NSPR = 0  
 MTP = 40  
 NDIM = 10000  
 MBD = 72  
 NSSTPS = 1  
 NDSTPS = 1000  
 NRTH = 3  
 NGITMAX = 100  
 NLITMAX = 50  
 NOUT = 1

DT = 0.50000E-02  
 TIMEOSTF = 0.40000E+01  
 BETA = 0.25000E+00  
 GAMMA = 0.50000E+00  
 A0 = 0.12566E+00  
 A1 = 0.20372E-02  
 AGRV = 0.38640E+03  
 STFFACPHE = 0.10000E-01  
 STFFACFE = 0.20000E-01  
 STFFACPZ0 = 0.10000E-01  
 STFFACPZ1 = 0.10000E-01  
 TOLF = 0.10000E-04  
 TOLM = 0.10000E-04  
 TOLFIBELF = 0.10000E-04  
 TOLFIBELM = 0.10000E-04  
 THRESH = 0.20000E+03  
 NTHR = 2  
 ITHR = 3

## NODAL POINT INPUT

NODE	COORDX	COORDY	COORDZ	IDX	IDY	IDZ	IDTHX	IDTHY	IDTHZ	FX	FY	FZ	MX	MY	MZ	IDASSCOL
1	0.00000E+00	0.00000E+00	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
2	0.20000E+03	0.00000E+00	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
3	0.40000E+03	0.00000E+00	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
4	0.00000E+00	0.20000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
5	0.20000E+03	0.20000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
6	0.40000E+03	0.20000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
7	0.00000E+00	0.40000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
8	0.20000E+03	0.40000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
9	0.40000E+03	0.40000E+03	0.00000E+00	0	0	0	0	0	0	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0
10	0.00000E+00	0.00000E+00	0.12000E+03	1	1	1	2	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	1	
11	0.20000E+03	0.00000E+00	0.12000E+03	1	1	1	2	1	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	3	
12	0.40000E+03	0.00000E+00	0.12000E+03	1	1	1	2	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	5	
13	0.00000E+00	0.20000E+03	0.12000E+03	1	1	1	2	1	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	7	
14	0.20000E+03	0.20000E+03	0.12000E+03	1	1	1	1	1	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0	
15	0.40000E+03	0.20000E+03	0.12000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	11	
16	0.00000E+00	0.40000E+03	0.12000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	9	
17	0.20000E+03	0.40000E+03	0.12000E+03	1	1	1	1	1	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0	
18	0.40000E+03	0.40000E+03	0.12000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	13	
19	0.00000E+00	0.00000E+00	0.24000E+03	1	1	1	2	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	2	
20	0.20000E+03	0.00000E+00	0.24000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	4	
21	0.40000E+03	0.00000E+00	0.24000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	6	
22	0.00000E+00	0.20000E+03	0.24000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	8	
23	0.20000E+03	0.20000E+03	0.24000E+03	1	1	1	1	1	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0	
24	0.40000E+03	0.20000E+03	0.24000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	12	
25	0.00000E+00	0.40000E+03	0.24000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	10	
26	0.20000E+03	0.40000E+03	0.24000E+03	1	1	1	1	1	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	0	
27	0.40000E+03	0.40000E+03	0.24000E+03	1	1	1	1	2	0.00000E+00	0.00000E+00	-0.10000E+03	0.10000E+03	0.10000E+03	0.10000E+03	14	

Figure 6.13: 2-Story Building: Output file, *for006*



\*\*\*\*\*

NDOF = 126

\*\*\*\*\*

#### DEGREE OF FREEDOM NUMBERS

NODE	DOFX	DOFY	DOFZ	DOFRX	DOFRYB	DOFRYC	DOFRZB	DOFRZC
1	0	0	0	0	0	0	0	0
2	0	0	0	0	0	0	0	0
3	0	0	0	0	0	0	0	0
4	0	0	0	0	0	0	0	0
5	0	0	0	0	0	0	0	0
6	0	0	0	0	0	0	0	0
7	0	0	0	0	0	0	0	0
8	0	0	0	0	0	0	0	0
9	0	0	0	0	0	0	0	0
10	1	2	3	4	5	6	7	8
11	9	10	11	12	13	14	15	15
12	16	17	18	19	20	21	22	23
13	24	25	26	27	28	29	30	30
14	31	32	33	34	35	35	36	36
15	37	38	39	40	41	42	43	43
16	44	45	46	47	48	49	50	50
17	51	52	53	54	55	55	56	56
18	57	58	59	60	61	62	63	63
19	64	65	66	67	68	69	70	71
20	72	73	74	75	76	77	78	78
21	79	80	81	82	83	84	85	86
22	87	88	89	90	91	92	93	93
23	94	95	96	97	98	98	99	99
24	100	101	102	103	104	105	106	106
25	107	108	109	110	111	112	113	113
26	114	115	116	117	118	118	119	119
27	120	121	122	123	124	125	126	126

NOTE THAT DOFX,Y,& Z REFER TO GLOBAL AXES  
WHILE DOFRX,RYB,RYC,RZB,RZC ALL REFER TO PANEL ZONE  
LOCAL AXES.

IF, HOWEVER, IDASSCOL OF A PARTICULAR NODE IS ZERO,  
THEN THERE CANNOT BE ANY PANEL ZONES ASSOCIATED  
WITH THAT NODE. HENCE, DOFRYC & DOFRZC ARE MEANINGLESS  
AND THE ROTATIONS NOW REFER TO THE GLOBAL AXES  
IN THIS CASE, i.e., THE PANEL ZONE COORDINATE SYSTEM  
IS SYNONYMOUS WITH THE GLOBAL COORDINATE SYSTEM.

FOR GRAVITY COLUMNS THAT DO NOT HAVE ANY BEAMS  
FRAMING INTO THEM, SPECIFY IDASSCOL=0.

FOR GRAVITY COLUMNS THAT HAVE GRAVITY BEAMS FRAMING  
INTO THEM, SPECIFY NON-ZERO IDASSCOL. IN THIS CASE  
CLEAR LENGTH OF COLUMN WILL BE TAKEN FOR STIFFNESS  
COMPUTATIONS.

#### ELASTIC SPRING DATA

SPRING#	NODE#	DOF#	SPRING STIFFNESS
---------	-------	------	------------------

Figure 6.14: 2-Story Building: Output file, *for006* (Contd.)

JOINT DIMENSIONS

NODE	H	D	W
1	0.0000	0.0000	0.0000
2	0.0000	0.0000	0.0000
3	0.0000	0.0000	0.0000
4	0.0000	0.0000	0.0000
5	0.0000	0.0000	0.0000
6	0.0000	0.0000	0.0000
7	0.0000	0.0000	0.0000
8	0.0000	0.0000	0.0000
9	0.0000	0.0000	0.0000
10	24.4800	14.9800	15.5650
11	24.4800	14.9800	15.5650
12	24.4800	14.9800	15.5650
13	24.4800	14.9800	15.5650
14	0.0000	0.0000	0.0000
15	24.4800	14.9800	15.5650
16	24.4800	14.9800	15.5650
17	0.0000	0.0000	0.0000
18	24.4800	14.9800	15.5650
19	24.4800	14.9800	15.5650
20	24.4800	14.9800	15.5650
21	24.4800	14.9800	15.5650
22	24.4800	14.9800	15.5650
23	0.0000	0.0000	0.0000
24	24.4800	14.9800	15.5650
25	24.4800	14.9800	15.5650
26	0.0000	0.0000	0.0000
27	24.4800	14.9800	15.5650

ELEMENT INPUT

FIBER ELEMENT AREA REDUCTION CATEGORY INFORMATION

ICAT	IDSEG	IDFIB	FAFRAC
1	1	1	2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01
1	2	1	2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.10000E+01

ISEED = 999287

FIBER ELEMENT FRACTURE STRESS CATEGORY INFORMATION

ICAT	IDSEG	IDFIB	FYFRAC(1) TO (10)
1	1	1	2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10
1	2	1	2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10 0.90000E+10

ELEM	NODE1	NIATTACH	NODE2	N2ATTACH	IBC	IDES	ALPHA	EE	HSR	SIGY	FAC1	FAC2	LENGTH	IELTYPE	OLFSRAT	ES	SIGU	EPSS	EPSU	ICATFAFRAC	ICATFYFRAC
1	1	0	10	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	107.7600	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
2	10	5	19	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	95.5200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
3	2	0	11	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	107.7600	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
4	11	5	20	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	95.5200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
5	3	0	12	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	107.7600	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
6	12	5	21	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	95.5200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
7	4	0	13	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	107.7600	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
8	13	5	22	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	95.5200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
9	7	0	16	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	107.7600	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
10	16	5	25	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	95.5200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
11	6	0	15	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	107.7600	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
12	15	5	24	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	95.5200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
13	9	0	18	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	107.7600	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
14	18	5	27	6	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	95.5200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
15	5	0	14	0	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	120.0000	0	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
16	14	0	23	0	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	120.0000	0	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
17	8	0	17	0	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	120.0000	0	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
18	17	0	26	0	2	14159	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	120.0000	0	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
19	10	1	11	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
20	19	1	20	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
21	11	1	12	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
22	20	1	21	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
23	10	4	13	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	184.7275	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
24	19	4	22	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	184.7275	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
25	13	1	16	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
26	22	1	25	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
27	12	4	15	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	184.7275	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
28	21	4	24	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	184.7275	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
29	15	1	18	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	
30	24	1	27	2	1	24131	0.029000E+05	0.0250	0.50000E+02	1.0000	1.0000	185.0200	1	0.03	0.58000E+03	0.65000E+02	0.012	0.160	1	1	

Figure 6.15: 2-Story Building: Output file, *for006* (Contd.)

```

*****

NPZ = 18

*****

PANEL ZONE INPUT

PZ# NODE Z OR Y IPZTYPE GG HSH TAUY THK DTHK HPZ
1 10 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
2 10 2 1 0.11600E+05 0.0500 0.28900E+02 2.3800 0.0000 24.4800
3 11 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
4 12 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
5 12 2 1 0.11600E+05 0.0500 0.28900E+02 2.3800 0.0000 24.4800
6 13 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
7 15 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
8 16 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
9 18 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
10 19 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
11 19 2 1 0.11600E+05 0.0500 0.28900E+02 2.3800 0.0000 24.4800
12 20 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
13 21 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
14 21 2 1 0.11600E+05 0.0500 0.28900E+02 2.3800 0.0000 24.4800
15 22 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
16 24 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
17 25 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800
18 27 1 1 0.11600E+05 0.0500 0.28900E+02 0.7450 0.0000 24.4800

PLANE STRESS ELEMENT DATA

NO.OF PLANE STRESS ELEMENTS, NUMPLSTREL = 8
NO.OF PLANE STRESS ELEMENT MATERIALS, NUMPLSTRMAT = 1
NO.OF INTEGRATION POINTS FOR GAUSS QUADRATURE,
NINTPTPLSTR = 2
NO.OF PLANE STRESS ELEMENTS, NUMPLSTRSS, FOR WHICH
PRINCIPAL STRESS HISTORY NEEDS TO BE WRITTEN OUT
IN FILE PLSTRSS = 0

MATERIAL SETS
SET E POIS
1 0.36000E+04 0.30000E+00

PLANE STRESS ELEMENT THICKNESSES & MATERIAL SET NUMBERS AND CONNECTIVITY MATRIX
ELEM MATSET THK NODE1 NODE2 NODE3 NODE4
1 1 3.500 10 11 14 13
2 1 3.500 11 12 15 14
3 1 3.500 13 14 17 16
4 1 3.500 14 15 18 17
5 1 3.500 19 20 23 22
6 1 3.500 20 21 24 23
7 1 3.500 22 23 26 25
8 1 3.500 23 24 27 26

PLANE STRESS ELEMENTS FOR WHICH PRINCIPAL STRESS
OUTPUT IS REQUIRED (FILE PLSTRSS)

*****

MAX. BANDWIDTH INPUT BY USER = 72
MAX. BANDWIDTH FOR PLANE STRESS ELEMENTS = 36
MAX. BANDWIDTH FOR FRAME ELEMENTS = 71
USE THE GREATER OF THE ABOVE TWO BANDWIDTHS

*****

RESPONSE IDRTH (1) (2) (3) (4) (5) (6) (7) (8) (9) (10)
R 1 27 1 0 0 0 0 0 0 0 0
R 2 27 2 0 0 0 0 0 0 0 0
R 3 27 3 0 0 0 0 0 0 0 0

```

Figure 6.16: 2-Story Building: Output file, *for006* (Contd.)

```

          STATIC SOLUTION

AVE. NO. OF MEMBER ITER :   2
MAX. NO. OF MEMBER ITER :   2

DATE 06/20/2003   TIME 19:44:05

TIME = 0.005   NITER = 1
      R 1    R 2    R 3    R 4    R 5    R 6    R 7    R 8
0.30731E-16 -0.19949E-16 -0.23100E-01

          STATIC ELEMENT RESPONSES

DATE 06/20/2003   TIME 19:44:05

ELEM  MPYL1  MYL1  MPYL2  MYL2  MPZL1  MZL1  MPZL2  MZL2  KAPYL1  KAPYL2  KAPZL1  KAPZL2  PY  P  PEPS  TORSION,MXL  ALPHA  ELEM
1  0.14350E+05 -0.27200E-13 0.14350E+05 -0.83512E-13 0.73000E+04 -0.38395E-15 0.73000E+04 -0.67345E-15 0.51352E-21 0.42082E-20 -0.29931E-20 -0.83226E-21 0.23215E+04 -0.20000E+03 0.00000E+00 0.10972E-16 -0.14748E-18 1
2  0.14350E+05 0.11289E-12 0.14350E+05 0.10991E-13 0.73000E+04 -0.36491E-13 0.73000E+04 -0.58525E-13 0.78972E-20 0.24612E-20 -0.57439E-20 -0.43226E-20 0.23215E+04 -0.10000E+03 0.00000E+00 0.34036E-16 -0.70046E-18 2
3  0.14350E+05 -0.83508E-13 0.14350E+05 -0.28006E-13 0.73000E+04 0.14267E-13 0.73000E+04 -0.14501E-13 -0.12765E-20 -0.19639E-21 -0.19739E-20 0.24662E-20 0.23215E+04 -0.20000E+03 0.00000E+00 0.11284E-16 -0.15167E-18 3
4  0.14350E+05 0.14226E-12 0.14350E+05 0.15353E-15 0.73000E+04 0.14270E-13 0.73000E+04 0.62312E-14 0.30718E-20 -0.25505E-20 -0.19906E-20 0.26005E-21 0.23215E+04 -0.10000E+03 0.00000E+00 0.31725E-16 -0.68131E-18 4
5  0.14350E+05 0.27808E-13 0.14350E+05 -0.54200E-15 0.73000E+04 0.13548E-13 0.73000E+04 -0.15333E-13 -0.18090E-20 -0.95238E-22 -0.49150E-20 -0.16807E-20 0.23215E+04 -0.20000E+03 0.00000E+00 0.13298E-16 -0.17874E-18 5
6  0.14350E+05 -0.18044E-13 0.14350E+05 -0.27739E-12 0.73000E+04 -0.79680E-13 0.73000E+04 -0.59507E-13 -0.11179E-20 -0.11898E-19 -0.92865E-20 -0.66269E-20 0.23215E+04 -0.10000E+03 0.00000E+00 0.27540E-16 -0.68559E-18 6
7  0.14350E+05 0.30531E-13 0.14350E+05 0.94480E-13 0.73000E+04 0.14154E-13 0.73000E+04 -0.14900E-13 -0.40953E-20 0.12175E-19 -0.40993E-20 0.34307E-20 0.23215E+04 -0.20000E+03 0.00000E+00 0.10499E-16 0.90000E+02 7
8  0.14350E+05 0.18312E-12 0.14350E+05 0.38207E-12 0.73000E+04 -0.69268E-14 0.73000E+04 0.57000E-14 0.12205E-19 0.18489E-19 -0.26053E-20 0.16764E-21 0.23215E+04 -0.10000E+03 0.00000E+00 0.33323E-16 0.90000E+02 8
9  0.14350E+05 -0.31360E-12 0.14350E+05 -0.29378E-13 0.73000E+04 0.14196E-13 0.73000E+04 -0.14928E-13 -0.76959E-20 0.40328E-20 -0.44021E-20 0.41365E-20 0.23215E+04 -0.20000E+03 0.00000E+00 0.10473E-16 0.90000E+02 9
10 0.14350E+05 -0.14264E-12 0.14350E+05 -0.36646E-12 0.73000E+04 -0.35355E-13 0.73000E+04 -0.87272E-14 -0.78419E-20 -0.18100E-19 -0.31972E-20 0.23232E-21 0.23215E+04 -0.10000E+03 0.00000E+00 0.35885E-16 0.90000E+02 10
11 0.14350E+05 0.31242E-13 0.14350E+05 0.13072E-12 0.73000E+04 0.14158E-13 0.73000E+04 0.14158E-13 -0.40875E-20 0.14891E-19 -0.40848E-20 0.35616E-20 0.23215E+04 -0.20000E+03 0.00000E+00 0.13233E-16 0.90000E+02 11
12 0.14350E+05 0.18336E-12 0.14350E+05 0.39632E-12 0.73000E+04 -0.74005E-14 0.73000E+04 0.68104E-14 0.12523E-19 0.18976E-19 -0.27176E-20 0.14402E-21 0.23215E+04 -0.10000E+03 0.00000E+00 0.25123E-16 0.90000E+02 12
13 0.14350E+05 -0.44135E-12 0.14350E+05 -0.29234E-13 0.73000E+04 0.14116E-13 0.73000E+04 -0.14306E-13 -0.83564E-20 0.52441E-20 -0.47582E-20 0.38202E-20 0.23215E+04 -0.20000E+03 0.00000E+00 0.11925E-16 0.90000E+02 13
14 0.14350E+05 -0.13304E-12 0.14350E+05 -0.17923E-12 0.73000E+04 -0.35774E-13 0.73000E+04 -0.24722E-13 -0.60796E-20 -0.11392E-19 -0.29644E-20 0.80813E-21 0.23215E+04 -0.10000E+03 0.00000E+00 0.41397E-16 0.90000E+02 14
15 0.14350E+05 -0.39121E-13 0.14350E+05 0.27953E-13 0.73000E+04 -0.30004E-13 0.73000E+04 0.15059E-13 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.23215E+04 -0.20000E+03 0.00000E+00 0.00000E+00 0.90000E+02 15
16 0.14350E+05 -0.28060E-13 0.14350E+05 0.98560E-15 0.73000E+04 -0.15245E-13 0.73000E+04 0.13953E-15 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.23215E+04 -0.10000E+03 0.00000E+00 0.00000E+00 0.90000E+02 16
17 0.14350E+05 -0.38312E-13 0.14350E+05 0.27951E-13 0.73000E+04 -0.33188E-13 0.73000E+04 0.18134E-13 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.23215E+04 -0.20000E+03 0.00000E+00 0.00000E+00 0.90000E+02 17
18 0.14350E+05 -0.28052E-13 0.14350E+05 0.55364E-14 0.73000E+04 -0.18345E-13 0.73000E+04 -0.24556E-15 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.23215E+04 -0.10000E+03 0.00000E+00 0.00000E+00 0.90000E+02 18
19 0.18500E+05 -0.51337E-12 0.18500E+05 0.12832E-11 0.40750E+04 0.45795E-16 0.40750E+04 -0.12223E-15 -0.57769E-20 0.40879E-19 -0.35222E-23 -0.12473E-22 0.19250E+04 -0.69823E-15 0.00000E+00 -0.21464E-16 0.31766E-17 19
20 0.18500E+05 0.37127E-13 0.18500E+05 -0.94078E-12 0.40750E+04 0.15751E-15 0.40750E+04 -0.10522E-15 0.20836E-19 -0.65912E-20 0.15574E-22 0.65499E-22 0.19250E+04 0.94719E-15 0.00000E+00 -0.28951E-16 0.50467E-17 20
21 0.18500E+05 -0.28017E-12 0.18500E+05 0.12454E-11 0.40750E+04 0.10554E-15 0.40750E+04 -0.17686E-15 -0.15348E-19 0.36927E-19 0.36104E-23 -0.54249E-22 0.19250E+04 -0.11084E-14 0.00000E+00 0.87718E-17 0.37872E-17 21
22 0.18500E+05 0.20669E-12 0.18500E+05 -0.37883E-12 0.40750E+04 0.27246E-15 0.40750E+04 -0.79534E-16 0.90472E-20 -0.57594E-20 0.26987E-22 0.75731E-22 0.19250E+04 0.11164E-14 0.00000E+00 0.27739E-17 0.63057E-17 22
23 0.18500E+05 0.79928E-13 0.18500E+05 0.34715E-13 0.40750E+04 0.60232E-16 0.40750E+04 -0.17135E-15 -0.15057E-19 0.41043E-20 -0.14215E-22 -0.62551E-24 0.19250E+04 0.48350E-15 0.00000E+00 -0.38032E-16 0.53184E-17 23
24 0.18500E+05 0.25530E-12 0.18500E+05 -0.12813E-12 0.40750E+04 0.16617E-15 0.40750E+04 -0.28891E-15 -0.55938E-20 -0.20102E-19 -0.20677E-23 0.29691E-22 0.19250E+04 -0.81912E-15 0.00000E+00 -0.53736E-16 0.73688E-17 24
25 0.18500E+05 -0.25857E-12 0.18500E+05 0.56951E-12 0.40750E+04 0.14428E-15 0.40750E+04 -0.22023E-15 -0.10232E-19 0.83011E-20 -0.15397E-22 -0.14640E-22 0.19250E+04 -0.25247E-14 0.00000E+00 -0.99256E-17 0.76222E-17 25
26 0.18500E+05 -0.7602E-12 0.18500E+05 -0.15233E-13 0.40750E+04 0.28110E-15 0.40750E+04 -0.57559E-15 -0.10837E-19 -0.38136E-20 -0.34955E-22 -0.99482E-22 0.19250E+04 0.23710E-14 0.00000E+00 -0.16605E-16 0.10748E-16 26
27 0.18500E+05 0.49573E-12 0.18500E+05 -0.47691E-12 0.40750E+04 0.24441E-15 0.40750E+04 -0.88323E-16 0.60446E-20 -0.18561E-19 0.44591E-22 0.46452E-22 0.19250E+04 0.84333E-15 0.00000E+00 -0.33387E-17 0.70004E-17 27
28 0.18500E+05 -0.21874E-12 0.18500E+05 -0.26885E-12 0.40750E+04 0.42739E-15 0.40750E+04 -0.27543E-15 -0.27172E-19 -0.25529E-19 -0.24725E-22 0.38743E-22 0.19250E+04 -0.14540E-14 0.00000E+00 0.39612E-16 0.11743E-16 28
29 0.18500E+05 -0.83845E-12 0.18500E+05 0.10150E-11 0.40750E+04 0.72466E-16 0.40750E+04 -0.22679E-15 -0.16023E-19 0.31575E-19 -0.56188E-22 -0.18607E-22 0.19250E+04 -0.18096E-14 0.00000E+00 -0.90735E-17 0.75972E-17 29
30 0.18500E+05 -0.32722E-12 0.18500E+05 0.48837E-13 0.40750E+04 0.31601E-15 0.40750E+04 -0.10813E-14 -0.6179E-20 0.41691E-20 -0.87473E-23 -0.38588E-21 0.19250E+04 0.17989E-14 0.00000E+00 -0.17941E-16 0.10704E-16 30

NODE NPZZORY MY M PGAM
10 1 0.63164E+04 -0.20808E-12 0.00000E+00
12 2 0.20966E+05 -0.28582E-13 0.00000E+00
11 1 0.63164E+04 -0.27118E-12 0.12037E-34
12 1 0.63164E+04 -0.22922E-13 0.00000E+00
12 2 0.20966E+05 -0.98635E-13 0.00000E+00
13 1 0.63164E+04 0.12668E-12 0.60185E-35
15 1 0.63164E+04 0.40725E-13 0.15046E-35
16 1 0.63164E+04 0.58048E-13 0.00000E+00
18 1 0.63164E+04 0.11164E-13 0.00000E+00
19 1 0.63164E+04 -0.28985E-12 0.00000E+00
19 2 0.20966E+05 -0.59698E-13 0.00000E+00
20 1 0.63164E+04 -0.44631E-12 0.00000E+00
21 1 0.63164E+04 -0.46878E-13 0.00000E+00
21 2 0.20966E+05 -0.15971E-12 0.00000E+00
22 1 0.63164E+04 0.24300E-12 0.12037E-34
24 1 0.63164E+04 0.25168E-12 0.00000E+00
25 1 0.63164E+04 0.64624E-13 -0.30093E-35
27 1 0.63164E+04 -0.52931E-13 0.00000E+00

```

Figure 6.17: 2-Story Building: Output file, *for006* (Contd.)

```

ELEM YLMSTRESS ZLMSTRESS PSTRESS TSTRESS
 1 0.32879E-15 0.70005E-17 0.43076E+01 0.43076E+01
 2 0.44446E-15 0.60837E-15 0.21538E+01 0.21538E+01
 3 0.33720E-15 0.15074E-15 0.43076E+01 0.43076E+01
 4 0.56009E-15 0.14834E-15 0.21538E+01 0.21538E+01
 5 0.10976E-15 0.15938E-15 0.43076E+01 0.43076E+01
 6 0.10921E-14 0.82827E-15 0.21538E+01 0.21538E+01
 7 0.37197E-15 0.15489E-15 0.43076E+01 0.43076E+01
 8 0.15042E-14 0.72005E-16 0.21538E+01 0.21538E+01
 9 0.12346E-14 0.15518E-15 0.43076E+01 0.43076E+01
10 0.14428E-14 0.36751E-15 0.21538E+01 0.21538E+01
11 0.51464E-15 0.14717E-15 0.43076E+01 0.43076E+01
12 0.15611E-14 0.76928E-16 0.21538E+01 0.21538E+01
13 0.17376E-14 0.14871E-15 0.43076E+01 0.43076E+01
14 0.70562E-15 0.37187E-15 0.21538E+01 0.21538E+01
15 0.15402E-15 0.31189E-15 0.43076E+01 0.43076E+01
16 0.11047E-15 0.15847E-15 0.21538E+01 0.21538E+01
17 0.15083E-15 0.34499E-15 0.43076E+01 0.43076E+01
18 0.11044E-15 0.19070E-15 0.21538E+01 0.21538E+01
19 0.39002E-14 0.23063E-17 0.18136E-16 0.39207E-14
20 0.28595E-14 0.29718E-17 0.24602E-16 0.28871E-14
21 0.37856E-14 0.33370E-17 0.28790E-16 0.38177E-14
22 0.11515E-14 0.51407E-17 0.28996E-16 0.11856E-14
23 0.24294E-15 0.32329E-17 0.12558E-16 0.25873E-15
24 0.77600E-15 0.54511E-17 0.21276E-16 0.80273E-15
25 0.17310E-14 0.41553E-17 0.65577E-16 0.18008E-14
26 0.23283E-14 0.10860E-16 0.61584E-16 0.24008E-14
27 0.15068E-14 0.46115E-17 0.21905E-16 0.15333E-14
28 0.81718E-15 0.80639E-17 0.37767E-16 0.86301E-15
29 0.30852E-14 0.42790E-17 0.47002E-16 0.31365E-14
30 0.99459E-15 0.20402E-16 0.46723E-16 0.10617E-14

NODE XDISP YDISP ZDISP
 1 0.00000E+00 0.00000E+00 0.00000E+00
 2 0.00000E+00 0.00000E+00 0.00000E+00
 3 0.00000E+00 0.00000E+00 0.00000E+00
 4 0.00000E+00 0.00000E+00 0.00000E+00
 5 0.00000E+00 0.00000E+00 0.00000E+00
 6 0.00000E+00 0.00000E+00 0.00000E+00
 7 0.00000E+00 0.00000E+00 0.00000E+00
 8 0.00000E+00 0.00000E+00 0.00000E+00
 9 0.00000E+00 0.00000E+00 0.00000E+00
10 0.74394E-17 -0.36494E-17 -0.16006E-01
11 0.73238E-17 -0.46873E-17 -0.16006E-01
12 0.71402E-17 -0.58130E-17 -0.16006E-01
13 0.84428E-17 -0.35694E-17 -0.16006E-01
14 0.84328E-17 -0.47312E-17 -0.17824E-01
15 0.84333E-17 -0.56734E-17 -0.16006E-01
16 0.94119E-17 -0.39878E-17 -0.16006E-01
17 0.94896E-17 -0.46542E-17 -0.17824E-01
18 0.95750E-17 -0.59733E-17 -0.16006E-01
19 0.23211E-16 -0.12509E-16 -0.23100E-01
20 0.23368E-16 -0.16335E-16 -0.23100E-01
21 0.23553E-16 -0.20007E-16 -0.23100E-01
22 0.27042E-16 -0.12645E-16 -0.23100E-01
23 0.27004E-16 -0.16235E-16 -0.26737E-01
24 0.26990E-16 -0.20247E-16 -0.23100E-01
25 0.30836E-16 -0.12251E-16 -0.23100E-01
26 0.30817E-16 -0.16215E-16 -0.26737E-01
27 0.30731E-16 -0.19949E-16 -0.23100E-01

```

Figure 6.18: 2-Story Building: Output file, *for006* (Contd.)

```

TIME HISTORY INPUT
NDSTPS = 1000 DT = 5.000000000000D-03

TIME HISTORY SCALING FACTORS

GAMULTX GAMULTY GAMULTZ
386.4000 386.4000 386.4000

SCALED X GROUND ACCELERATION SPACED AT DT
0.0000 2.6426 5.2852 7.9278 10.5700 9.5858 8.6020 7.6179 6.6337 5.9324
5.2311 4.5298 3.8286 3.8110 3.7934 3.7758 3.7582 3.9069 4.0560 4.2048
4.3536 3.0861 1.8187 0.5513 -0.7161 0.5117 1.7394 2.9671 4.1948 5.2477
6.3003 7.3532 8.4057 6.3683 4.3304 2.2928 0.2551 -0.4361 -1.1273 -1.8186
-2.5098 -1.1714 0.1671 1.5055 2.8439 3.1352 3.4264 3.7177 4.0089 3.4434
.
.
.
.

SCALED Y GROUND ACCELERATION SPACED AT DT
0.0000 2.7024 5.4046 8.1071 10.8095 9.3049 7.8006 6.2960 4.7917 3.6156
2.4396 1.2635 0.0875 -0.4822 -1.0518 -1.6215 -2.1911 -2.3798 -2.5684 -2.7571
-2.9457 -4.7118 -6.4776 -8.2435 -10.0093 -8.4181 -6.8269 -5.2353 -3.6441 -3.2422
-2.8402 -2.4382 -2.0363 -2.6935 -3.3507 -4.0077 -4.6650 -5.7098 -6.7547 -7.7995
-8.8443 -7.5657 -6.2875 -5.0089 -3.7305 -3.1314 -2.5324 -1.9334 -1.3344 -2.3694
.
.
.
.

SCALED Z GROUND ACCELERATION SPACED AT DT
0.0000 -2.4571 -4.9142 -7.3714 -9.8285 -8.9761 -8.1233 -7.2705 -6.4181 -4.8798
-3.3417 -1.8035 -0.2653 -1.5429 -2.8204 -4.0982 -5.3756 -5.7817 -6.1882 -6.5943
-7.0004 -5.9312 -4.8617 -3.7925 -2.7232 -3.6239 -4.5247 -5.4254 -6.3261 -4.7489
-3.1715 -1.5941 -0.0168 -1.4324 -2.8480 -4.2635 -5.6793 -4.9911 -4.3030 -3.6149
-2.9268 -3.0216 -3.1164 -3.2111 -3.3058 -2.2805 -1.2552 -0.2299 0.7954 2.0045

DYNAMIC SOLUTION

DATE 06/20/2003 TIME 19:44:05

TIME = 0.010 NITER = 0 ZT = 0.00000E+00
R 1 R 2 R 3 R 4 R 5 R 6 R 7 R 8
0.30731E-16 -0.19949E-16 -0.23100E-01

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 06/20/2003 TIME 19:44:05

TIME = 0.015 NITER = 2 ZT = 0.00000E+00
R 1 R 2 R 3 R 4 R 5 R 6 R 7 R 8
-0.16540E-04 -0.16939E-04 -0.23087E-01

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 06/20/2003 TIME 19:44:05

TIME = 0.020 NITER = 4 ZT = 0.00000E+00
R 1 R 2 R 3 R 4 R 5 R 6 R 7 R 8
-0.99321E-04 -0.10178E-03 -0.23024E-01

```

Figure 6.19: 2-Story Building: Output file, *for006* (Contd.)

```

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 4

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 06/20/2003 TIME 19:44:05

TIME = 0.025  NITER = 4  ZT = 0.00000E+00
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
-0.31488E-03 -0.32290E-03 -0.22887E-01
  •
  •
  •
DATE 06/20/2003 TIME 19:51:54

TIME = 5.000  NITER = 20  ZT = 0.00000E+00
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
0.36033E+01 0.55247E-02 -0.55692E-01

AVE. NO.OF MEMBER ITER : 6
MAX. NO.OF MEMBER ITER : 12

AVE. NO.OF MEMBER ITER : 5
MAX. NO.OF MEMBER ITER : 10

AVE. NO.OF MEMBER ITER : 4
MAX. NO.OF MEMBER ITER : 9

AVE. NO.OF MEMBER ITER : 4
MAX. NO.OF MEMBER ITER : 8

AVE. NO.OF MEMBER ITER : 4
MAX. NO.OF MEMBER ITER : 8

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 8

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 7

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 7

AVE. NO.OF MEMBER ITER : 3
MAX. NO.OF MEMBER ITER : 6

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 6

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 5

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 5

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 4

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 4

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 3

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

AVE. NO.OF MEMBER ITER : 2
MAX. NO.OF MEMBER ITER : 2

DATE 06/20/2003 TIME 19:51:56

TIME = 5.005  NITER = 20  ZT = 0.00000E+00
  R 1  R 2  R 3  R 4  R 5  R 6  R 7  R 8
0.38069E+01 -0.88064E-01 -0.57891E-01

```

Figure 6.20: 2-Story Building: Output file, *for006* (Contd.)

MINIMUM STATIC PLUS DYNAMIC ELEMENT RESPONSES

DATE 06/20/2003 TIME 19:51:56

ELEM	MPYL1	MYL1	MPYL2	MYL2	MPZL1	MZL1	MPZL2	MZL2	KAPYL1	KAPYL2	KAPZL1	KAPZL2	PY	P	PEPS	TORSION/MXL	ALPHA	ELEM
1	0.14350E+05	-0.91422E+04	0.14350E+05	-0.31924E+04	0.73000E+04	-0.31255E+04	0.73000E+04	-0.21229E+04	-0.13482E-03	-0.17217E-05	-0.26018E-03	-0.64681E-05	0.23215E+04	-0.45441E+03	-0.54210E-19	-0.41923E+01	-0.52241E-01	1
2	0.14350E+05	-0.48970E+04	0.14350E+05	-0.64187E+04	0.73000E+04	-0.36004E+04	0.73000E+04	-0.38956E+04	-0.63757E-05	-0.33742E-04	-0.19765E-04	-0.82464E-04	0.23215E+04	-0.21081E+03	-0.54210E-19	-0.71147E+01	-0.18102E+00	2
3	0.14350E+05	-0.99613E+04	0.14350E+05	-0.38489E+04	0.73000E+04	-0.54430E+03	0.73000E+04	-0.19786E+03	-0.55520E-05	-0.21037E-05	-0.17364E-05	-0.70693E-06	0.23215E+04	-0.22859E+03	-0.54210E-19	-0.45357E+01	-0.51167E-01	3
4	0.14350E+05	-0.63859E+04	0.14350E+05	-0.86239E+04	0.73000E+04	-0.82915E+03	0.73000E+04	-0.32161E+02	-0.31260E-05	-0.42479E-05	-0.23399E-05	-0.20004E-06	0.23215E+04	-0.11524E+03	-0.54210E-19	-0.65518E+01	-0.17332E+00	4
5	0.14350E+05	-0.92977E+04	0.14350E+05	-0.32081E+04	0.73000E+04	-0.10600E+04	0.73000E+04	-0.71121E+03	-0.38437E-04	-0.17491E-05	-0.33550E-05	-0.22127E-05	0.23215E+04	-0.43208E+03	-0.54210E-19	-0.45784E+01	-0.44724E-01	5
6	0.14350E+05	-0.48776E+04	0.14350E+05	-0.64323E+04	0.73000E+04	-0.13162E+04	0.73000E+04	-0.11605E+04	-0.23957E-05	-0.21299E-04	-0.36166E-05	-0.32207E-05	0.23215E+04	-0.20484E+03	-0.54210E-19	-0.70326E+01	-0.16950E+00	6
7	0.14350E+05	-0.64211E+04	0.14350E+05	-0.30769E+04	0.73000E+04	-0.45325E+04	0.73000E+04	-0.61154E+03	-0.15126E-03	-0.17038E-05	-0.21646E-02	-0.20278E-05	0.23215E+04	-0.23016E+03	-0.54210E-19	-0.44018E+01	0.89955E+02	7
8	0.14350E+05	-0.55802E+04	0.14350E+05	-0.64657E+04	0.73000E+04	-0.51804E+03	0.73000E+04	-0.72096E+02	-0.23737E-05	-0.31797E-05	-0.15187E-05	-0.34498E-06	0.23215E+04	-0.11617E+03	-0.54210E-19	-0.72947E+01	0.89831E+02	8
9	0.14350E+05	-0.54825E+04	0.14350E+05	-0.20147E+04	0.73000E+04	-0.54057E+04	0.73000E+04	-0.70954E+03	-0.41879E-03	-0.11055E-05	-0.30598E-02	-0.23805E-05	0.23215E+04	-0.38150E+03	-0.54210E-19	-0.46149E+01	0.89954E+02	9
10	0.14350E+05	-0.40040E+04	0.14350E+05	-0.45520E+04	0.73000E+04	-0.48959E+03	0.73000E+04	-0.67633E+02	-0.19881E-05	-0.22486E-05	-0.14343E-05	-0.30958E-06	0.23215E+04	-0.17498E+03	-0.54210E-19	-0.68775E+01	0.89837E+02	10
11	0.14350E+05	-0.21377E+04	0.14350E+05	-0.10664E+04	0.73000E+04	-0.59253E+04	0.73000E+04	-0.61486E+03	-0.12112E-05	-0.61686E-06	-0.33498E-05	-0.20538E-05	0.23215E+04	-0.23021E+03	-0.54210E-19	-0.42526E+01	0.89954E+02	11
12	0.14350E+05	-0.32725E+04	0.14350E+05	-0.19849E+04	0.73000E+04	-0.54027E+03	0.73000E+04	-0.71120E+02	-0.16199E-05	-0.98677E-06	-0.15852E-05	-0.33624E-06	0.23215E+04	-0.11432E+03	-0.54210E-19	-0.72493E+01	0.89828E+02	12
13	0.14350E+05	-0.19416E+04	0.14350E+05	-0.70423E+03	0.73000E+04	-0.69701E+04	0.73000E+04	-0.67888E+03	-0.11211E-05	-0.41656E-06	-0.49221E-03	-0.23053E-05	0.23215E+04	-0.34956E+03	-0.54210E-19	-0.42798E+01	0.89954E+02	13
14	0.14350E+05	-0.20828E+04	0.14350E+05	-0.13427E+04	0.73000E+04	-0.55654E+03	0.73000E+04	-0.61603E+02	-0.10470E-05	-0.69167E-06	-0.15515E-05	-0.30861E-06	0.23215E+04	-0.16735E+03	-0.54210E-19	-0.68412E+01	0.89832E+02	14
15	0.14350E+05	-0.11394E+04	0.14350E+05	-0.38626E+03	0.73000E+04	-0.58085E+04	0.73000E+04	-0.53003E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.22779E+03	-0.16263E-18	-0.18631E+01	0.89955E+02	15
16	0.14350E+05	-0.11990E+04	0.14350E+05	-0.57206E+00	0.73000E+04	-0.46496E+03	0.73000E+04	-0.21385E-01	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.11448E+03	-0.67763E-18	-0.24499E-01	0.89911E+02	16
17	0.14350E+05	-0.11360E+04	0.14350E+05	-0.38421E+03	0.73000E+04	-0.72736E+04	0.73000E+04	-0.42956E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.22694E+03	-0.16263E-18	-0.25839E+01	0.89935E+02	17
18	0.14350E+05	-0.11779E+04	0.14350E+05	-0.10977E+01	0.73000E+04	-0.52535E+03	0.73000E+04	-0.34101E-01	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.11428E+03	-0.54210E-19	-0.50669E-01	0.89872E+02	18
19	0.18500E+05	-0.96991E+04	0.18500E+05	-0.59088E+04	0.40750E+04	-0.25127E+02	0.40750E+04	-0.11184E+02	-0.50011E-05	-0.30118E-05	-0.79538E-06	-0.65436E-06	0.19250E+04	-0.23215E+01	-0.54210E-19	-0.49008E+00	-0.23310E+00	19
20	0.18500E+05	-0.73713E+04	0.18500E+05	-0.47537E+04	0.40750E+04	-0.39682E+02	0.40750E+04	-0.18827E+02	-0.37941E-05	-0.24282E-05	-0.91626E-06	-0.63890E-06	0.19250E+04	-0.14130E+02	-0.54210E-19	-0.35310E+00	-0.15602E+00	20
21	0.18500E+05	-0.59581E+04	0.18500E+05	-0.97489E+04	0.40750E+04	-0.10285E+02	0.40750E+04	-0.56463E+01	-0.30463E-05	-0.50269E-05	-0.61042E-06	-0.55494E-06	0.19250E+04	-0.15340E+01	-0.54210E-19	-0.24989E+01	-0.10854E+00	21
22	0.18500E+05	-0.47970E+04	0.18500E+05	-0.73832E+04	0.40750E+04	-0.24104E+01	0.40750E+04	-0.41189E+01	-0.24459E-05	-0.38170E-05	-0.56390E-06	-0.57991E-06	0.19250E+04	-0.10424E+02	-0.54210E-19	-0.20028E+01	-0.63203E-01	22
23	0.18500E+05	-0.64254E+04	0.18500E+05	-0.46392E+04	0.40750E+04	-0.37314E+02	0.40750E+04	-0.16479E+02	-0.33063E-05	-0.23629E-05	-0.76156E-06	-0.56449E-06	0.19250E+04	-0.93093E+01	-0.54210E-19	-0.47469E+01	0.58736E+00	23
24	0.18500E+05	-0.45004E+04	0.18500E+05	-0.33770E+04	0.40750E+04	-0.34809E+02	0.40750E+04	-0.17415E+02	-0.23275E-05	-0.17340E-05	-0.84216E-06	-0.65124E-06	0.19250E+04	-0.51787E+01	-0.54210E-19	-0.53250E+01	0.52024E+00	24
25	0.18500E+05	-0.49057E+04	0.18500E+05	-0.66572E+04	0.40750E+04	-0.12972E+01	0.40750E+04	-0.48873E+01	-0.25192E-05	-0.34168E-05	-0.55281E-06	-0.53878E-06	0.19250E+04	-0.73627E+01	-0.54210E-19	-0.24536E+01	0.93331E+00	25
26	0.18500E+05	-0.39276E+04	0.18500E+05	-0.52620E+04	0.40750E+04	-0.58036E+01	0.40750E+04	-0.36755E+01	-0.20215E-05	-0.27119E-05	-0.56283E-06	-0.53700E-06	0.19250E+04	-0.15553E+02	-0.54210E-19	-0.28049E+01	0.91083E+00	26
27	0.18500E+05	-0.20446E+04	0.18500E+05	-0.15287E+04	0.40750E+04	-0.14515E+01	0.40750E+04	-0.13340E+01	-0.10798E-06	-0.80881E-06	-0.55171E-06	-0.55428E-06	0.19250E+04	-0.11554E+02	-0.54210E-19	-0.49770E+01	0.60828E+00	27
28	0.18500E+05	-0.13406E+04	0.18500E+05	-0.10260E+04	0.40750E+04	-0.66623E+01	0.40750E+04	-0.85211E+01	-0.71237E-06	-0.54856E-06	-0.56728E-06	-0.55966E-06	0.19250E+04	-0.45967E+01	-0.54210E-19	-0.50258E+01	0.51830E+00	28
29	0.18500E+05	-0.15994E+04	0.18500E+05	-0.21220E+04	0.40750E+04	-0.98222E+01	0.40750E+04	-0.10461E+02	-0.83785E-06	-0.11174E-05	-0.54382E-06	-0.54848E-06	0.19250E+04	-0.50550E+01	-0.54210E-19	-0.25082E+01	0.96794E+00	29
30	0.18500E+05	-0.14033E+04	0.18500E+05	-0.16503E+04	0.40750E+04	-0.10741E+02	0.40750E+04	-0.19800E+02	-0.73620E-06	-0.86472E-06	-0.57393E-06	-0.68470E-06	0.19250E+04	-0.10281E+02	-0.54210E-19	-0.28742E+01	0.89798E+00	30

NODE	NPZZORY	MY	M	PGAM
10	1	0.63164E+04	-0.79730E+04	-0.14970E-02
11	2	0.20966E+05	-0.76138E+04	-0.67763E-20
11	1	0.63164E+04	-0.97539E+04	-0.47667E-02
12	1	0.63164E+04	-0.80533E+04	-0.16704E-02
12	2	0.20966E+05	-0.65492E+04	-0.54210E-19
13	1	0.63164E+04	-0.86727E+04	-0.32142E-02
15	1	0.63164E+04	-0.80253E+04	-0.18633E-02
16	1	0.63164E+04	-0.72405E+04	-0.54228E-03
18	1	0.63164E+04	-0.63015E+04	-0.67763E-20
19	1	0.63164E+04	-0.70048E+04	-0.30069E-03
19	2	0.20966E+05	-0.49822E+04	-0.54210E-19
20	1	0.63164E+04	-0.86822E+04	-0.24966E-02
21	1	0.63164E+04	-0.72503E+04	-0.55391E-03
21	2	0.20966E+05	-0.42740E+04	-0.23717E-19
22	1	0.63164E+04	-0.77730E+04	-0.13513E-02
24	1	0.63164E+04	-0.68074E+04	-0.15288E-03
25	1	0.63164E+04	-0.64207E+04	-0.68945E-05
27	1	0.63164E+04	-0.49879E+04	-0.65052E-18

Figure 6.21: 2-Story Building: Output file, *for006* (Contd.)



MAXIMUM STATIC PLUS DYNAMIC ELEMENT RESPONSES

DATE 06/20/2003 TIME 19:51:56

ELEM	MPYL1	MYL1	MPYL2	MYL2	MPZL1	MZL1	MPZL2	MZL2	KAPYL1	KAPYL2	KAPZL1	KAPZL2	PY	P	PEPS	TORSION.MXL	ALPHA	ELEM
1	0.14350E+05	0.10053E+05	0.14350E+05	0.36773E+04	0.73000E+04	0.39519E+04	0.73000E+04	0.31300E+04	0.17223E-02	0.20025E-05	0.24632E-02	0.95280E-05	0.23215E+04	0.97137E+02	0.13553E-19	0.38680E+01	-0.52241E-01	1
2	0.14350E+05	0.56184E+04	0.14350E+05	0.76542E+04	0.73000E+04	0.50759E+04	0.73000E+04	0.53685E+04	0.11106E-03	0.72711E-03	0.25420E-03	0.13065E-02	0.23215E+04	0.28470E+02	0.13553E-19	0.64460E+01	-0.18102E+00	2
3	0.14350E+05	0.12461E+05	0.14350E+05	0.49841E+04	0.73000E+04	0.25613E+04	0.73000E+04	0.57138E+03	0.60915E-03	0.27354E-05	0.88649E-03	0.17340E-05	0.23215E+04	-0.17834E+03	0.13553E-19	0.38096E+01	-0.51167E-01	3
4	0.14350E+05	0.74110E+04	0.14350E+05	0.10372E+05	0.73000E+04	0.24275E+03	0.73000E+04	0.97631E+02	0.36047E-05	0.50835E-05	0.74010E-06	0.42500E-06	0.23215E+04	-0.87920E+02	0.13553E-19	0.60257E+01	-0.17332E+00	4
5	0.14350E+05	0.12094E+05	0.14350E+05	0.38870E+04	0.73000E+04	0.48142E+04	0.73000E+04	0.34117E+04	0.28195E-03	0.20978E-05	0.40188E-03	0.10365E-04	0.23215E+04	0.51953E+02	0.13553E-19	0.33457E+01	-0.44724E-01	5
6	0.14350E+05	0.54422E+04	0.14350E+05	0.80921E+04	0.73000E+04	0.40413E+04	0.73000E+04	0.46669E+04	0.26554E-05	0.81032E-06	0.10885E-04	0.59688E-04	0.23215E+04	0.11648E+02	0.13553E-19	0.67526E+01	-0.16950E+00	6
7	0.14350E+05	0.89847E+04	0.14350E+05	0.37899E+04	0.73000E+04	0.46178E+04	0.73000E+04	0.28686E+03	0.13532E-02	0.21031E-05	0.32609E-03	0.11787E-05	0.23215E+04	-0.17359E+03	0.13553E-19	0.33212E+01	0.89955E+02	7
8	0.14350E+05	0.66237E+04	0.14350E+05	0.88996E+04	0.73000E+04	0.63892E+03	0.73000E+04	0.54322E+02	0.32187E-05	0.43586E-05	0.18478E-05	0.28969E-06	0.23215E+04	-0.80857E+02	0.13553E-19	0.67630E+01	0.89831E+02	8
9	0.14350E+05	0.79253E+04	0.14350E+05	0.28410E+04	0.73000E+04	0.59498E+04	0.73000E+04	0.64903E+03	0.17493E-02	0.15588E-05	0.99177E-03	0.22661E-05	0.23215E+04	-0.76960E+02	0.13553E-19	0.33977E+01	0.89954E+02	9
10	0.14350E+05	0.51310E+04	0.14350E+05	0.69754E+04	0.73000E+04	0.60201E+03	0.73000E+04	0.53271E+02	0.25173E-05	0.34288E-05	0.17083E-05	0.26635E-06	0.23215E+04	-0.44984E+02	0.13553E-19	0.60999E+01	0.89837E+02	10
11	0.14350E+05	0.94356E+04	0.14350E+05	0.45333E+04	0.73000E+04	0.47358E+04	0.73000E+04	0.22014E+03	0.13014E-03	0.24846E-05	0.27128E-03	0.78639E-06	0.23215E+04	-0.17711E+03	0.13553E-19	0.34380E+01	0.89954E+02	11
12	0.14350E+05	0.54159E+04	0.14350E+05	0.76262E+04	0.73000E+04	0.64063E+03	0.73000E+04	0.68093E+02	0.26402E-03	0.37504E-05	0.18436E-05	0.32559E-06	0.23215E+04	-0.86077E+02	0.13553E-19	0.68274E+01	0.89828E+02	12
13	0.14350E+05	0.83009E+04	0.14350E+05	0.37928E+04	0.73000E+04	0.61259E+04	0.73000E+04	0.42016E+03	0.46650E-03	0.20878E-05	0.94692E-03	0.17029E-05	0.23215E+04	-0.15380E+03	0.13553E-19	0.34083E+01	0.89954E+02	13
14	0.14350E+05	0.38627E+04	0.14350E+05	0.54155E+04	0.73000E+04	0.57111E+03	0.73000E+04	0.67379E+02	0.18874E-05	0.26798E-05	0.16237E-05	0.31270E-06	0.23215E+04	-0.78850E+02	0.13553E-19	0.63865E+01	0.89832E+02	14
15	0.14350E+05	0.55253E+04	0.14350E+05	0.11994E+04	0.73000E+04	0.47522E+04	0.73000E+04	0.46505E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.17814E+03	0.54210E-19	0.30094E+01	0.89955E+02	15
16	0.14350E+05	0.38626E+03	0.14350E+05	0.18241E+01	0.73000E+04	0.51921E+03	0.73000E+04	0.22833E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.87783E+02	0.40658E-19	0.15838E-01	0.89911E+02	16
17	0.14350E+05	0.55011E+04	0.14350E+05	0.11781E+04	0.73000E+04	0.64769E+04	0.73000E+04	0.52555E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.17814E+03	0.21684E-18	0.43532E-01	0.89935E+02	17
18	0.14350E+05	0.38421E+03	0.14350E+05	0.29037E+01	0.73000E+04	0.42824E+03	0.73000E+04	0.33816E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.23215E+04	-0.87287E+02	0.13553E-19	0.44543E-01	0.89872E+02	18
19	0.18500E+05	0.11535E+05	0.18500E+05	0.70743E+04	0.40750E+04	0.44898E+01	0.40750E+04	0.36133E+01	0.59328E-05	0.35720E-05	0.54973E-06	0.56016E-06	0.19250E+04	0.11702E+02	0.13553E-19	0.21012E+01	-0.23310E+00	19
20	0.18500E+05	0.88716E+04	0.18500E+05	0.55863E+04	0.40750E+04	0.51844E+01	0.40750E+04	0.18591E+01	0.45608E-05	0.28353E-05	0.57265E-06	0.57199E-06	0.19250E+04	0.90702E+01	0.13553E-19	0.14629E+01	-0.15602E+00	20
21	0.18500E+05	0.73213E+04	0.18500E+05	0.11875E+05	0.40750E+04	0.11235E+02	0.40750E+04	0.15328E+02	0.37049E-05	0.60968E-05	0.55776E-06	0.59651E-06	0.19250E+04	0.60535E+01	0.13553E-19	0.15844E+01	-0.10854E+00	21
22	0.18500E+05	0.58530E+04	0.18500E+05	0.92055E+04	0.40750E+04	0.11788E+02	0.40750E+04	0.26144E+02	0.29797E-05	0.47395E-05	0.60610E-06	0.82106E-06	0.19250E+04	0.17574E+02	0.13553E-19	0.11804E+01	-0.63203E-01	22
23	0.18500E+05	0.97858E+04	0.18500E+05	0.59787E+04	0.40750E+04	0.34039E+01	0.40750E+04	0.31647E+01	0.50422E-05	0.30403E-05	0.55922E-06	0.56235E-06	0.19250E+04	0.41717E+01	0.13553E-19	0.44123E+01	0.58736E+00	23
24	0.18500E+05	0.63356E+04	0.18500E+05	0.44928E+04	0.40750E+04	0.13422E+02	0.40750E+04	0.11422E+02	0.32646E-05	0.23007E-05	0.56169E-06	0.55385E-06	0.19250E+04	0.17406E+02	0.13553E-19	0.47162E+01	0.52024E+00	24
25	0.18500E+05	0.63407E+04	0.18500E+05	0.10117E+05	0.40750E+04	0.16225E+02	0.40750E+04	0.14659E+02	0.32338E-05	0.52012E-05	0.56102E-06	0.59960E-06	0.19250E+04	0.96686E+01	0.13553E-19	0.28562E+01	0.93331E+00	25
26	0.18500E+05	0.54136E+04	0.18500E+05	0.79767E+04	0.40750E+04	0.16365E+02	0.40750E+04	0.24337E+02	0.27684E-05	0.41005E-05	0.60511E-06	0.70018E-06	0.19250E+04	0.20924E+02	0.13553E-19	0.29613E+01	0.91083E+00	26
27	0.18500E+05	0.85375E+04	0.18500E+05	0.54960E+04	0.40750E+04	0.29202E+02	0.40750E+04	0.14246E+02	0.43899E-05	0.27959E-05	0.81157E-06	0.64469E-06	0.19250E+04	0.42080E+01	0.13553E-19	0.44317E+01	0.60828E+00	27
28	0.18500E+05	0.53891E+04	0.18500E+05	0.40261E+04	0.40750E+04	0.27290E+02	0.40750E+04	0.14752E+02	0.27799E-05	0.20575E-05	0.67597E-06	0.60868E-06	0.19250E+04	0.18356E+02	0.13553E-19	0.46760E+01	0.51836E+00	28
29	0.18500E+05	0.56584E+04	0.18500E+05	0.87169E+04	0.40750E+04	0.66680E+00	0.40750E+04	0.44350E+00	0.28867E-05	0.45015E-05	0.37894E-06	0.25624E-06	0.19250E+04	0.50121E+01	0.13553E-19	0.28665E+01	0.96794E+00	29
30	0.18500E+05	0.44487E+04	0.18500E+05	0.61652E+04	0.40750E+04	0.37575E+01	0.40750E+04	0.57990E+01	0.22831E-05	0.31736E-05	0.57052E-06	0.53176E-06	0.19250E+04	0.21335E+02	0.13553E-19	0.29562E+01	0.99798E+00	30

NODE	NPZZORY	MY	M	PGAM
10	1	0.63164E+04	0.69481E+04	0.25314E-03
10	2	0.20966E+05	0.49500E+04	0.43368E-18
11	1	0.63164E+04	0.84475E+04	0.29072E-02
12	1	0.63164E+04	0.69501E+04	0.25470E-03
12	2	0.20966E+05	0.15700E+04	0.10842E-18
13	1	0.63164E+04	0.70559E+04	0.34704E-03
15	1	0.63164E+04	0.24807E+04	0.10842E-18
16	1	0.63164E+04	0.48378E+04	0.21684E-18
18	1	0.63164E+04	0.15285E+04	0.19516E-17
19	1	0.63164E+04	0.59356E+04	0.21684E-18
19	2	0.20966E+05	0.35540E+04	0.10842E-18
20	1	0.63164E+04	0.76275E+04	0.10939E-02
21	1	0.63164E+04	0.59399E+04	0.24395E-18
21	2	0.20966E+05	0.10607E+04	0.31171E-18
22	1	0.63164E+04	0.57743E+04	0.13010E-17
24	1	0.63164E+04	0.17674E+04	0.65052E-18
25	1	0.63164E+04	0.41799E+04	0.43368E-18
27	1	0.63164E+04	0.12340E+04	0.27105E-19

Figure 6.22: 2-Story Building: Output file, *for006* (Contd.)

DATE 06/20/2003 TIME 19:51:56

## PANEL ZONE INELASTIC ROTATIONS

NODE	YRPZ	ZRPZ
1	0.00	0.00
2	0.00	0.00
3	0.00	0.00
4	0.00	0.00
5	0.00	0.00
6	0.00	0.00
7	0.00	0.00
8	0.00	0.00
9	0.00	0.00
10	0.00	0.15
11	0.00	0.48
12	0.00	0.17
13	0.00	0.32
14	0.00	0.00
15	0.00	0.19
16	0.00	0.05
17	0.00	0.00
18	0.00	0.00
19	0.00	0.03
20	0.00	0.25
21	0.00	0.06
22	0.00	0.14
23	0.00	0.00
24	0.00	0.02
25	0.00	0.00
26	0.00	0.00
27	0.00	0.00

## BEAM/COLUMN END INELASTIC ROTATIONS

ELEM	N1	N1ATTACH	YROT1	ZROT1	N2	N2ATTACH	YROT2	ZROT2
1	1	0	0.17	0.25	10	6	0.00	0.00
2	10	5	0.01	0.03	19	6	0.07	0.13
3	2	0	0.06	0.09	11	6	0.00	0.00
4	11	5	0.00	0.00	20	6	0.00	0.00
5	3	0	0.03	0.04	12	6	0.00	0.00
6	12	5	0.00	0.00	21	6	0.00	0.01
7	4	0	0.14	0.22	13	6	0.00	0.00
8	13	5	0.00	0.00	22	6	0.00	0.00
9	7	0	0.17	0.31	16	6	0.00	0.00
10	16	5	0.00	0.00	25	6	0.00	0.00
11	6	0	0.01	0.03	15	6	0.00	0.00
12	15	5	0.00	0.00	24	6	0.00	0.00
13	9	0	0.05	0.09	18	6	0.00	0.00
14	18	5	0.00	0.00	27	6	0.00	0.00
15	5	0	0.00	0.00	14	0	0.00	0.00
16	14	0	0.00	0.00	23	0	0.00	0.00
17	8	0	0.00	0.00	17	0	0.00	0.00
18	17	0	0.00	0.00	26	0	0.00	0.00
19	10	1	0.00	0.00	11	2	0.00	0.00
20	19	1	0.00	0.00	20	2	0.00	0.00
21	11	1	0.00	0.00	12	2	0.00	0.00
22	20	1	0.00	0.00	21	2	0.00	0.00
23	10	4	0.00	0.00	13	2	0.00	0.00
24	19	4	0.00	0.00	22	2	0.00	0.00
25	13	1	0.00	0.00	16	2	0.00	0.00
26	22	1	0.00	0.00	25	2	0.00	0.00
27	12	4	0.00	0.00	15	2	0.00	0.00
28	21	4	0.00	0.00	24	2	0.00	0.00
29	15	1	0.00	0.00	18	2	0.00	0.00
30	24	1	0.00	0.00	27	2	0.00	0.00

## PRINCIPAL STRESSES AND MAXIMUM SHEAR STRESS IN PLANE STRESS ELEMENTS

ELEM. NO.	MAX. SIGMA1	ANGLE	MIN. SIGMA2	ANGLE	MAX. SHEAR STRESS
1	0.12679886	45.99663587	-0.15288483	45.99663587	0.13984185
2	0.09927425	-44.89577728	-0.10511986	-46.39223013	0.10218357
3	0.07395474	38.57957058	-0.10974282	38.57957058	0.09184878
4	0.07023142	-48.03172168	-0.09353122	34.99353682	0.06192345
5	0.23955270	-45.83219764	-0.31261173	-45.83219764	0.27608222
6	0.20445100	44.90472800	-0.22223180	44.90911577	0.21329384
7	0.16577347	47.38731017	-0.19988095	-50.66346429	0.16959102
8	0.13964590	-54.35553228	-0.16048637	44.02047764	0.13520314

Figure 6.23: 2-Story Building: Output file, *for006* (Contd.)

## Appendix A Glossary of Input & Output Variables

Variable	Input File	Data Block	Description
<i>A0</i>	<i>for005</i>	Integration & Iteration Data	Rayleigh damping parameter.
<i>A1</i>	<i>for005</i>	Integration & Iteration Data	Rayleigh damping parameter.
<i>AGRAV</i>	<i>for005</i>	Integration & Iteration Data	Acceleration due to gravity.
<i>ALPHA</i>	<i>for005</i>	Beam Element Data	Element orientation angle.
<i>ALPHA</i>	<i>for006</i>	Static and Dynamic Element Responses	Updated orientation angle of the element.
<i>AREA</i>	<i>for009</i>	Beam Section Data	Area of the cross-section.
<i>BETA</i>	<i>for005</i>	Integration & Iteration Data	Newmark integration parameter.
<i>DEPTH</i>	<i>for009</i>	Beam Section Data	Section depth.
<i>DOF</i>	<i>for005</i>	Spring Element Data	The degree-of-freedom (1-8) the motion of which is resisted by the spring.
<i>DT</i>	<i>for005</i>	Integration & Iteration Data	Time increment for dynamic time-history analysis.
<i>DTHK</i>	<i>for005</i>	Panel Zone Element Data	Doubler plate thickness.
<i>DTOUT</i>	<i>for005</i>	Deformed Shape Output Control Data	Time interval at which deformed shape nodal coordinates are required.
<i>E</i>	<i>for005</i>	Diaphragm Element Material Data	Elastic modulus.
<i>EE</i>	<i>for005</i>	Beam Element Data	Modulus of Elasticity.
<i>ELEM</i>	<i>for005</i>	Time-History Output Control Data	Beam element number.
<i>ELEM</i>	<i>for005</i>	Time-History Output Control Data	Elastofiber type beam element number.
<i>ELEM</i>	<i>for006</i>	Static and Dynamic Element Responses	Beam element number.
<i>ELEM</i>	<i>for006</i>	Static Element Responses	Beam element number.
<i>EPSS</i>	<i>for005</i>	Beam Element Data	Fiber strain at onset of strain-hardening in an elastofiber element.
<i>EPSU</i>	<i>for005</i>	Beam Element Data	Fiber ultimate strain in an elastofiber element.
<i>ES</i>	<i>for005</i>	Beam Element Data	Fiber tangent modulus (slope of stress-strain curve) in an elastofiber element at onset of strain-hardening.
<i>FAC1</i>	<i>for005</i>	Beam Element Data	Degree of continuity at end <i>i</i> .
<i>FAC2</i>	<i>for005</i>	Beam Element Data	Degree of continuity at end <i>j</i> .
<i>FAFRAC</i>	<i>for005</i>	Categories for Elastofiber	Fraction of fiber area to be used

Variable	Input File	Data Block	Description
<i>FIB</i>	<i>for005</i>	Element Fiber Area Reduction Time-History Output Control Data	in the analysis (0.0-1.0).  Fiber number.
<i>FLGWDTH</i>	<i>for009</i>	Beam Section Data	Flange width.
<i>FLGTHK</i>	<i>for009</i>	Beam Section Data	Flange thickness.
<i>FX</i>	<i>for005</i>	Nodal Data	Static nodal force in the global X direction.
<i>FY</i>	<i>for005</i>	Nodal Data	Static nodal force in the global Y direction.
<i>FYFRAC</i> (10)	<i>for005</i>	Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution	Fracture stress probability distribution.
<i>FZ</i>	<i>for005</i>	Nodal Data	Static nodal force in the global Z direction.
<i>G</i>	<i>for005</i>	Panel Zone Element Data	Shear modulus.
<i>GAMMA</i>	<i>for005</i>	Integration & Iteration Data	Newmark integration parameter.
<i>GAMULTX</i>	<i>for005</i>	Ground Acceleration Scaling Factor	Scaling factor for X-component of ground acceleration.
<i>GAMULTY</i>	<i>for005</i>	Ground Acceleration Scaling Factor	Scaling factor for Y-component of ground acceleration.
<i>GAMULTZ</i>	<i>for005</i>	Ground Acceleration Scaling Factor	Scaling factor for Z-component of ground acceleration.
<i>HPZ</i>	<i>for005</i>	Panel Zone Element Data	Panel zone height.
<i>HSH</i>	<i>for005</i>	Beam Element Data	Strain hardening modulus.
<i>HSH</i>	<i>for005</i>	Panel Zone Element Data	Strain hardening modulus.
<i>IBC</i>	<i>for005</i>	Beam Element Data	Beam or column identifier: 1 for Beam 2 for Column.
<i>ICAT</i>	<i>for005</i>	Categories for Elastofiber Element Fiber Area Reduction	Fiber area reduction category number.
<i>ICAT</i>	<i>for005</i>	Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution	Fiber fracture stress probability distribution category number.
<i>ICATFAFRAC</i>	<i>for005</i>	Beam Element Data	Category for fiber area reduction.
<i>ICATFYFRAC</i>	<i>for005</i>	Beam Element Data	Category for fiber fracture stress.
<i>IDASSCOL</i>	<i>for005</i>	Nodal Data	Element number of the column that is to be associated with the joint under consideration.
<i>IDES</i>	<i>for005</i>	Beam Element Data	Section designation.
<i>IDES</i>	<i>for009</i>	Beam Section Data	Section designation (label).
<i>IDFIB</i> (20)	<i>for005</i>	Categories for Elastofiber Element Fiber Area Reduction	Fiber numbers.

Variable	Input File	Data Block	Description
<i>IDFIB(20)</i>	<i>for005</i>	Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution	Fiber numbers.
<i>IDSEG</i>	<i>for005</i>	Categories for Elastofiber Element Fiber Area Reduction	Fiber segment 1 or 2.
<i>IDSEG</i>	<i>for005</i>	Categories for Elastofiber Element Fiber Fracture Stress Prob. Distribution	Fiber segment 1 or 2.
<i>IELTYPE</i>	<i>for005</i>	Beam Element Data	Element type: 0 for plastic-hinge, 1 for elastofiber.
<i>IPZTYPE</i>	<i>for005</i>	Panel Zone Element Data	Panel zone type: 0 for bilinear and 1 for linear-quadratic.
<i>ISEED</i>	<i>for005</i>	Seed for Random Number Generation for Elastofiber Element Fiber Fracture Stress Computation	Seed to generate random numbers.
<i>ITHR</i>	<i>for005</i>	Integration & Iteration Data	Threshold DOF for system picture.
<i>KAPYL1</i>	<i>for006</i>	Static and Dynamic Element Responses	Kink rotation in the element at node 1 about the local $Y'$ (major) axis.
<i>KAPYL2</i>	<i>for006</i>	Static and Dynamic Element Responses	Kink rotation in the element at node 2 about the local $Y'$ (major) axis.
<i>KAPZL1</i>	<i>for006</i>	Static and Dynamic Element Responses	Kink rotation in the element at node 1 about the local $Z'$ (minor) axis.
<i>KAPZL2</i>	<i>for006</i>	Static and Dynamic Element Responses	Kink rotation in the element at node 2 about the local $Z'$ (minor) axis.
<i>LENGTH</i>	<i>for006</i>	Beam Element Data	Clear length of the beam element between attachment points.
<i>LMG</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Increment to be applied to the node numbers for the next element in the series to be generated.
<i>M</i>	<i>for006</i>	Static and Dynamic Element Responses	Panel zone moment.
<i>MAT</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Material number.
<i>MBD</i>	<i>for005</i>	Main Control Data	Global half bandwidth.
<i>MPYL1</i>	<i>for006</i>	Static and Dynamic Element Responses	Plastic moment capacity at node 1 of the element about the local $Y'$ (major) axis.
<i>MPYL2</i>	<i>for006</i>	Static and Dynamic Element Responses	Plastic moment capacity at node 2 of the element about the local $Y'$ (major) axis.
<i>MPZL1</i>	<i>for006</i>	Static and Dynamic Element Responses	Plastic moment capacity at node 1 of the element about the local $Z'$ (minor) axis.
<i>MPZL2</i>	<i>for006</i>	Static and Dynamic Element Responses	Plastic moment capacity at node 2 of the element about the local $Z'$ (minor) axis.

Variable	Input File	Data Block	Description
<i>MTP</i>	<i>for005</i>	Main Control Data	Maximum number of turning points.
<i>MX</i>	<i>for005</i>	Nodal Data	Nodal mass in weight units in the global X direction.
<i>MXL</i>	<i>for006</i>	Static and Dynamic Element Responses	Torsion in the element.
<i>MY</i>	<i>for005</i>	Nodal Data	Nodal mass in weight units in the global Y direction.
<i>MY</i>	<i>for006</i>	Static and Dynamic Element Responses	Panel zone yield moment.
<i>MYL1</i>	<i>for006</i>	Static and Dynamic Element Responses	Moment in the element at node 1 about the local Y' (major) axis.
<i>MYL2</i>	<i>for006</i>	Static and Dynamic Element Responses	Moment in the element at node 2 about the local Y' (major) axis.
<i>MZ</i>	<i>for005</i>	Nodal Data	Nodal mass in weight units in the global Z direction.
<i>MZL1</i>	<i>for006</i>	Static and Dynamic Element Responses	Moment in the element at node 1 about the local Z' (minor) axis.
<i>MZL2</i>	<i>for006</i>	Static and Dynamic Element Responses	Moment in the element at node 2 about the local Z' (minor) axis.
<i>N</i>	<i>for005</i>	Diaphragm Element Connectivity Data	First element in the series to be generated.
<i>N1</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Node 1 of first element in the series to be generated.
<i>N1</i>	<i>for005</i>	Diaphragm Element Output Control Data	First diaphragm element in the series for which principal stress-strain histories are required.
<i>N1ATTACH</i>	<i>for006</i>	Beam Element Data	Attachment point of element at node 1.
<i>N2</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Node 2 of first element in the series to be generated.
<i>N2</i>	<i>for005</i>	Diaphragm Element Output Control Data	Last diaphragm element in the series for which principal stress-strain histories are required.
<i>N2ATTACH</i>	<i>for006</i>	Beam Element Data	Attachment point of element at node 2.
<i>N3</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Node 3 of first element in the series to be generated.
<i>N4</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Node 4 of first element in the series to be generated.
<i>NDIM</i>	<i>for005</i>	Main Control Data	Storage parameter for turning point locations.
<i>NDSTPS</i>	<i>for005</i>	Main Control Data	Number of dynamic analysis steps.
<i>NE</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Last element in the series to be generated.
<i>NFIBEL</i>	<i>for005</i>	Main Control Data	Number of elastofiber beam elements.

Variable	Input File	Data Block	Description
<i>NG</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Increment in diaphragm element number.
<i>NGITMAX</i>	<i>for005</i>	Main Control Data	Maximum number of global iterations per time step.
<i>NINTPTPLSTR</i>	<i>for005</i>	Diaphragm Element Control Data	Number of integration points for Gauss quadrature.
<i>NLITMAX</i>	<i>for005</i>	Main Control Data	Maximum number of local member (elastofiber element) iterations per time step.
<i>NNP</i>	<i>for005</i>	Main Control Data	Number of node points.
<i>NODE</i>	<i>for005</i>	Spring Element Data	Identity of the node-point at which the spring is to be located.
<i>NODE</i>	<i>for005</i>	Panel Zone Element Data	Node number.
<i>NODE</i>	<i>for005</i>	Time-History Output Control Data	Node number.
<i>NODE</i>	<i>for005</i>	Time-History Output Control Data	Node number.
<i>NODE</i>	<i>for006</i>	Static and Dynamic Element Responses	Node number.
<i>NODE1</i>	<i>for005</i>	Beam Element Data	Node number at end $i$ of the element.
<i>NODE2</i>	<i>for005</i>	Beam Element Data	Node number at end $j$ of the element.
<i>NOUT</i>	<i>for005</i>	Main Control Data	Output data is written for every $NOUT^{th}$ time step.
<i>NPHEL</i>	<i>for005</i>	Main Control Data	Number of plastic hinge beam elements.
<i>NPLSTRSS</i>	<i>for005</i>	Diaphragm Element Control Data	Number of diaphragm elements for which principal stress-strain history is required.
<i>NPZ0</i>	<i>for005</i>	Main Control Data	Number of panel zones with bilinear shear stress-strain relation.
<i>NPZ1</i>	<i>for005</i>	Main Control Data	Number of panel zones with linear-quadratic shear stress-strain relation.
<i>NPZZORY</i>	<i>for005</i>	Panel Zone Element Data	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
<i>NPZZORY</i>	<i>for006</i>	Static and Dynamic Element Responses	Panel zone orientation: 1 for $X'Z'$ plane and 2 for $X'Y'$ plane.
<i>NRTH</i>	<i>for005</i>	Main Control Data	Number of response time histories to be output.
<i>NSPR</i>	<i>for005</i>	Main Control Data	Number of translational and rotational springs.
<i>NSSTPS</i>	<i>for005</i>	Main Control Data	Number of static analysis steps.
<i>NTHR</i>	<i>for005</i>	Integration & Iteration Data	Threshold node for system picture.
<i>NUMPLSTREL</i>	<i>for005</i>	Diaphragm Element Control Data	Number of diaphragm elements.
<i>NUMPLSTRMAT</i>	<i>for005</i>	Diaphragm Element	Number of diaphragm element material

Variable	Input File	Data Block	Description
		Control Data	types.
<i>OLFSRATIO</i>	<i>for005</i>	Beam Element Data	Ratio of fiber segment length to element total length.
<i>P</i>	<i>for006</i>	Static and Dynamic Element Responses	Axial force in the element.
<i>PEPS</i>	<i>for006</i>	Static and Dynamic Element Responses	Axial strain in the element.
<i>PGAM</i>	<i>for006</i>	Static and Dynamic Element Responses	Plastic shear strain in the panel zone.
<i>POIS</i>	<i>for005</i>	Diaphragm Element Material Data	Poisson's ratio.
<i>PSTRESS</i>	<i>for006</i>	Static Element Responses	Axial stress in the element.
<i>PY</i>	<i>for006</i>	Static and Dynamic Element Responses	Axial yield force of the element.
<i>RESPID</i>	<i>for005</i>	Time-History Output Control Data	Nodal response quantity.
<i>RESPID</i>	<i>for005</i>	Time-History Output Control Data	Element response quantity.
<i>RESPID</i>	<i>for005</i>	Time-History Output Control Data	Panel zone response quantity.
<i>RESPID</i>	<i>for005</i>	Time-History Output Control Data	Fiber response quantity.
<i>RXRES</i>	<i>for005</i>	Nodal Data	Restraint to nodal rotation about joint local $\bar{X}$ axis.
<i>RYRES</i>	<i>for005</i>	Nodal Data	Restraint to nodal rotation about joint local $\bar{Y}$ axis.
<i>RZRES</i>	<i>for005</i>	Nodal Data	Restraint to nodal rotation about joint local $\bar{Z}$ axis.
<i>SEG</i>	<i>for005</i>	Time-History Output Control Data	Segment number.
<i>SIGU</i>	<i>for005</i>	Beam Element Data	Fiber ultimate stress in an elastofiber element.
<i>SIGY</i>	<i>for005</i>	Beam Element Data	Yield stress.
<i>SMI</i>	<i>for009</i>	Beam Section Data	Moment of inertia about the strong (major) axis of the section.
<i>SPRSTF</i>	<i>for005</i>	Spring Element Data	Spring stiffness.
<i>SSM</i>	<i>for009</i>	Beam Section Data	Section modulus about the strong (major) axis of the section.
<i>STFFACFE</i>	<i>for005</i>	Integration & Iteration Data	Fraction of elastic stiffness of elastofiber element to be added to its tangent stiffness for Newton-Raphson iteration.
<i>STFFACPHE</i>	<i>for005</i>	Integration & Iteration Data	Fraction of elastic stiffness of plastic hinge element to be added to its tangent stiffness for Newton-Raphson iteration.



Variable	Input File	Data Block	Description
<i>STFFACPZ0</i>	<i>for005</i>	Integration & Iteration Data	Fraction of elastic stiffness of bilinear panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
<i>STFFACPZ1</i>	<i>for005</i>	Integration & Iteration Data	Fraction of elastic stiffness of linear-quadratic panel zone element to be added to its tangent stiffness for Newton-Raphson iteration.
<i>SZ</i>	<i>for009</i>	Beam Section Data	Plastic section modulus about the strong (major) axis of the section.
<i>TAUY</i>	<i>for005</i>	Panel Zone Element Data	Shear yield stress.
<i>THK</i>	<i>for005</i>	Panel Zone Element Data	Panel zone thickness.
<i>THK</i>	<i>for005</i>	Diaphragm Element Connectivity Data	Diaphragm element thickness.
<i>THRESH</i>	<i>for005</i>	Integration & Iteration Data	Threshold value for system picture.
<i>TIMEOSTF</i>	<i>for005</i>	Integration & Iteration Data	Time till which the initial elastic global stiffness matrix is to be used for Newton-Raphson iteration.
<i>TOLF</i>	<i>for005</i>	Integration & Iteration Data	Force tolerance for global convergence of displacement DOF.
<i>TOLFIBELF</i>	<i>for005</i>	Integration & Iteration Data	Force tolerance for local convergence of displacement DOF of an elastofiber element.
<i>TOLFIBELM</i>	<i>for005</i>	Integration & Iteration Data	Moment tolerance for local convergence of rotation DOF of an elastofiber element.
<i>TOLM</i>	<i>for005</i>	Integration & Iteration Data	Moment tolerance for global convergence of rotation DOF.
<i>TORJ</i>	<i>for009</i>	Beam Section Data	Torsional constant.
<i>TSTRESS</i>	<i>for006</i>	Static Element Responses	Total stress in the element.
<i>WEBTHK</i>	<i>for009</i>	Beam Section Data	Web thickness.
<i>WMI</i>	<i>for009</i>	Beam Section Data	Moment of inertia about the weak (minor) axis of the section.
<i>WSM</i>	<i>for009</i>	Beam Section Data	Section modulus about the weak (minor) axis of the section.
<i>WZ</i>	<i>for009</i>	Beam Section Data	Plastic section modulus about the weak (minor) axis of the section.
<i>X</i>	<i>for005</i>	Nodal Data	X coordinate of node-point.
<i>XDISP</i>	<i>for006</i>	Static Element Responses	Displacement of the node in the global X direction.
<i>XRES</i>	<i>for005</i>	Nodal Data	Restraint to nodal translation in global X direction.
<i>Y</i>	<i>for005</i>	Nodal Data	Y coordinate of node-point.
<i>YDISP</i>	<i>for006</i>	Static Element Responses	Displacement of the node in the global

Variable	Input File	Data Block	Description
<i>YLMSTRESS</i>	<i>for006</i>	Static Element Responses	Y direction. Greater of the flexural stresses at the two ends of the element about the local $Y'$ (major) axis.
<i>YRES</i>	<i>for005</i>	Nodal Data	Restraint to nodal translation in global Y direction.
<i>YROT1</i>	<i>for006</i>	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 1 of element about the local $Y'$ (major) axis.
<i>YROT2</i>	<i>for006</i>	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 2 of element about the local $Y'$ (major) axis.
<i>YRPZ</i>	<i>for006</i>	Panel Zone Inelastic Rotations	Plastic rotation (%) in a $X'Y'$ panel zone.
<i>Z</i>	<i>for005</i>	Nodal Data	Z coordinate of node-point.
<i>ZDISP</i>	<i>for006</i>	Static Element Responses	Displacement of the node in the global Z direction.
<i>ZLMSTRESS</i>	<i>for006</i>	Static Element Responses	Greater of the flexural stresses at the two ends of the element about the local $Z'$ (minor) axis.
<i>ZRES</i>	<i>for005</i>	Nodal Data	Restraint to nodal translation in global Z direction.
<i>ZROT1</i>	<i>for006</i>	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 1 of element about the local $Z'$ (minor) axis.
<i>ZROT2</i>	<i>for006</i>	Beam Element End Inelastic Rotations	Plastic rotation (%) at node 2 of element about the local $Z'$ (minor) axis.
<i>ZRPZ</i>	<i>for006</i>	Panel Zone Inelastic Rotations	Plastic rotation (%) in a $X'Z'$ panel zone.

## Bibliography

- [1] BERTERO, V. V., POPOV, E. P., AND KRAWINKLER, H. Beam-column subassemblies under repeated loading. *Journal of the Structural Division, ASCE* 98, ST5 (1972), 1137–1159.
- [2] CHALLA, V. R. M. Nonlinear seismic behavior of steel planar moment-resisting frames. Tech. Rep. EERL 92-01, Earthquake Engineering Research Laboratory, California Institute of Technology, 1992.
- [3] COOK, R. D., MALKUS, D. S., AND PLESHA, M. E. *Concepts and Applications of Finite Element Analysis (Third Edition)*. John Wiley and Sons, 1989.
- [4] FIELDING, D. J., AND CHEN, W. F. Steel frame analysis and connection shear deformation. *Journal of the Structural Division, ASCE* 99, ST1 (1973), 1–18.
- [5] HALL, J. F., AND CHALLA, V. R. M. Beam-column modeling. *Journal of Engineering Mechanics* 121, 12 (1995), 1284–1291.
- [6] KASSIMALI, A. Large deformation analysis of elastic-plastic frames. *Journal of the Structural Division, ASCE* 109, 8 (1983), 1869–1886.
- [7] KATO, B. Beam-to-column connection research in japan. *Journal of the Structural Division, ASCE* 108, ST2 (1982), 343–360.
- [8] KRAWINKLER, H., BERTERO, V. V., AND POPOV, E. P. Shear behavior of steel frame joints. *Journal of the Structural Division, ASCE* 101, ST11 (1975), 2317–2336.
- [9] KRAWINKLER, H., AND POPOV, E. P. Shear behavior of moment connections and joints. *Journal of the Structural Division, ASCE* 108, ST2 (1982), 373–391.
- [10] KRISHNAN, S. Three-dimensional nonlinear analysis of tall irregular steel buildings subject to strong ground motion. Tech. Rep. EERL 2003-01, Earthquake Engineering Research Laboratory, California Institute of Technology, 2003.

- [11] KRISHNAN, S., AND HALL, J. F. Modeling steel frame buildings in three dimensions - part i: Panel zone and plastic hinge beam elements. Submitted to ASCE Journal of Engineering Mechanics in January 2004.
- [12] KRISHNAN, S., AND HALL, J. F. Modeling steel frame buildings in three dimensions - part ii: Elastofiber beam element and examples. Submitted to ASCE Journal of Engineering Mechanics in January 2004.
- [13] MATTIASSON, K. Numerical results from large deflection beam and frame problems analyzed by means of elliptic integrals. *International Journal for Numerical Methods in Engineering* 17, (1) (1980), 145–153.
- [14] ORAN, C. Tangent stiffness in plane frames. *Journal of the Structural Division, ASCE* 99, 6 (1973), 973–985.
- [15] ORAN, C. Tangent stiffness in space frames. *Journal of the Structural Division, ASCE* 99, 6 (1973), 987–1001.
- [16] PINKNEY, R. B. Cyclic plastic analysis of structural steel joints. Tech. Rep. UCB/EERC-73-15, Earthquake Engineering Research Center, University of California, Berkeley, 1973.
- [17] POPOV, E. P., AND PETERSSON, H. Cyclic metal plasticity: Experiments and theory. *Journal of the Engineering Mechanics Division, ASCE* 104, EM6 (1978), 1371–1388.